

AD-A245 957



NAVAL POSTGRADUATE SCHOOL

Monterey, California

2



DTIC  
ELECTE  
FEB 14 1992  
S D D

## THESIS

DYNAMIC MODELING AND MODAL ANALYSIS  
OF AN AIR-TO-AIR MISSILE

by

Michael Allen Shutty

September, 1991

Thesis Advisor:

Edward M. Wu

Approved for public release; distribution is unlimited

92-03668



92 2 12 141

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE				
1a REPORT SECURITY CLASSIFICATION Unclassified			1b RESTRICTIVE MARKINGS	
2a SECURITY CLASSIFICATION AUTHORITY			3 DISTRIBUTION/AVAILABILITY OF REPORT Approved for public release; distribution is unlimited.	
2b DECLASSIFICATION/DOWNGRADING SCHEDULE				
4 PERFORMING ORGANIZATION REPORT NUMBER(S)			5 MONITORING ORGANIZATION REPORT NUMBER(S)	
6a NAME OF PERFORMING ORGANIZATION Naval Postgraduate School		6b OFFICE SYMBOL (if applicable) 31		7a NAME OF MONITORING ORGANIZATION Naval Postgraduate School
6c ADDRESS (City, State, and ZIP Code) Monterey, CA 93943 5000			7b ADDRESS (City, State, and ZIP Code) Monterey, CA 93943 5000	
8a NAME OF FUNDING/SPONSORING ORGANIZATION		8b OFFICE SYMBOL (if applicable)		9 PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER
8c ADDRESS (City, State, and ZIP Code)			10 SOURCE OF FUNDING NUMBERS	
			Program Element No.	Project No.
			Task No.	Work Unit Accession Number
11 TITLE (Include Security Classification) DYNAMIC MODELING AND MODAL ANALYSIS OF AN AIR-TO-AIR MISSILE				
12 PERSONAL AUTHOR(S) Michael A. Shetty				
13a TYPE OF REPORT Master's Thesis		13b TIME COVERED From To		14 DATE OF REPORT (year, month, day) September 1991
				15 PAGE COUNT 112
16 SUPPLEMENTARY NOTATION The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.				
17 COSATI CODES			18 SUBJECT TERMS (continue on reverse if necessary and identify by block number)	
FIELD	GROUP	SUBGROUP	Vibration, modal analysis, finite element modeling, dynamic modeling, P-3, AIM-9	
19 ABSTRACT (continue on reverse if necessary and identify by block number) The P-3 Orion patrol airplane has a need for an air-to-air missile system for defense against enemy aircraft on its long range missions. In response to this need, the Naval Air Test Center was tasked in 1989 to conduct a P-3/AIM-9 (Sidewinder) integration program. In support of this program, a vibration test stand was established at NPS, and a ground vibration characterization was conducted to determine if a potential flutter problem existed. This test resulted in the development of a two-degree of freedom lumped-mass model and experimental determination of the missile's resonance modes in pitch. With the recent termination of the P-7A, the P-3 community is now looking to the P-3 Orion II program to carry it into the 21st century. The Orion II will most likely have a beefed up wing structure, necessitating an analysis of this wing in conjunction with the AIM-9 missile. This investigation responds to that requirement by concurrently developing a mathematical model of the AIM-9 missile using finite element techniques to analytically determine its modal parameters, and setting up a modal test system to quantify the parameters of this model by experimentally determining the missile's natural frequencies, mode shapes, and transient response. This fully instrumented test system and associated methodologies could then be the basis for conducting a complete modal test of the AIM-9 missile system, as well as to quantify the vibration characteristics of other candidate missile systems for the P-3 and its eventual successor.				
20 DISTRIBUTION/AVAILABILITY OF ABSTRACT <input checked="" type="checkbox"/> UNCLASSIFIED UNLIMITED <input type="checkbox"/> SAME AS REPORT <input type="checkbox"/> DTIC USERS			21 ABSTRACT SECURITY CLASSIFICATION Unclassified	
22a NAME OF RESPONSIBLE INDIVIDUAL Professor Edward M. Wu			22b TELEPHONE (Include Area code) (408) 646 3459	
			22c OFFICE SYMBOL AA/Wu	

DD FORM 1473, 84 MAR

85 APR edition may be used until exhausted  
All other editions are obsoleteSECURITY CLASSIFICATION OF THIS PAGE  
Unclassified

Approved for public release; distribution is unlimited.

Dynamic Modeling and Modal Analysis  
of an Air-to-Air Missile

by

Michael Allen Shutty  
B.S., Pennsylvania State University, 1983

Submitted in partial fulfillment  
of the requirements for the degree of

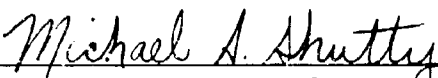
MASTER OF SCIENCE IN AERONAUTICAL ENGINEERING

from the

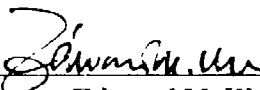
NAVAL POSTGRADUATE SCHOOL

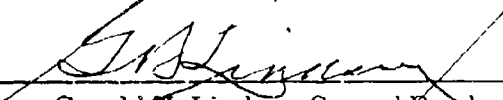
September 1991

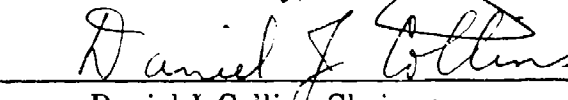
Author:

  
Michael A. Shutty

Approved by:

  
Edward M. Wu, Thesis Advisor

  
Gerald H. Lindsey, Second Reader

  
Daniel J. Collins, Chairman

Department of Aeronautics and Astronautics

## ABSTRACT

The P-3 Orion patrol airplane has a need for an air-to-air missile system for defense against enemy aircraft on its long-range missions. In response to this need, the Naval Air Test Center was tasked in 1989 to conduct a P-3/AIM-9 (Sidewinder) integration program. In support of this program, a vibration test stand was established at NPS, and a ground vibration characterization was conducted to determine if a potential flutter problem existed. This test resulted in the development of a two degree-of-freedom lumped-mass model and experimental determination of the missile's resonance modes in pitch. With the recent termination of the P-7A, the P-3 community is now looking at the Orion II program to carry it into the 21st century. The Orion II will most likely have a beefed-up wing structure, necessitating an analysis of this wing in conjunction with the AIM-9 missile. This investigation responds to that requirement by concurrently developing a mathematical model of the AIM-9 missile using finite element methods to analytically determine its modal parameters, and setting up a modal test system to quantify the parameters of this model by experimentally determining the missile's natural frequencies, mode shapes and transient response. This fully instrumented test system and associated methodologies could then be the basis for conducting a comprehensive modal test of the AIM-9 missile system, as well as to quantify the vibration characteristics of other candidate missile systems for the P-3 and its eventual successor.

Accession For	
NTIS CRA&I	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution	
Availability Codes	
Dist	Avail. and/or Special
A-1	

## TABLE OF CONTENTS

I.	INTRODUCTION . . . . .	1
A.	BACKGROUND . . . . .	1
B.	MISSILE MODELING TECHNIQUES . . . . .	5
C.	IMPORTANT DEFINITIONS . . . . .	6
II.	ANALYTICAL MODELING . . . . .	9
A.	FINITE ELEMENT METHOD BACKGROUND . . . . .	9
B.	MSC/PAL2 FINITE ELEMENT ANALYSIS SOFTWARE . . .	11
	1. Introduction . . . . .	11
	2. Basic Program Operation . . . . .	12
	a. File . . . . .	12
	b. Edit . . . . .	13
	c. Analysis . . . . .	14
	d. Graphics . . . . .	16
	e. Composite Capabilities . . . . .	17
	f. Advanced Capabilities . . . . .	17
	3. MSC/PAL2 Analysis Procedure . . . . .	18
	4. MSC/PAL2 Limitations . . . . .	20
C.	MSC-PAL2 SIDEWINDER MISSILE ANALYTICAL MODEL .	22
	1. Model Geometry . . . . .	22
	2. Determination of Modal Parameters . . . . .	26
	3. Modification of Modal Parameters . . . . .	29

III. EXPERIMENTAL MODAL ANALYSIS . . . . .	32
A. GENERAL . . . . .	32
B. PREVIOUS WORK ACCOMPLISHED . . . . .	35
1. Test Setup . . . . .	35
2. Previous Results . . . . .	36
C. MODAL TEST SETUP CONSIDERATIONS . . . . .	38
1. Transducer Selection . . . . .	38
2. Transducer Mounting . . . . .	40
3. Transducer Calibration . . . . .	41
4. Excitation Techniques . . . . .	41
a. Shaker Excitation . . . . .	41
b. Impulse (Impact) Excitation . . . . .	43
5. Specifying the Number of Degrees-of-Freedom . . . . .	44
D. SIDEWINDER MISSILE MODAL ANALYSIS TEST SETUP . . . . .	45
1. Test Structure . . . . .	45
2. Test Equipment . . . . .	46
E. MODAL-PC SOFTWARE BACKGROUND . . . . .	50
1. Introduction . . . . .	50
2. Major Features . . . . .	50
F. EMODAL-PC MODAL TESTING FUNDAMENTALS . . . . .	51
1. Test Procedures . . . . .	51
2. Modal Testing vs. Operating Deflection Shape Test . . . . .	52
a. Modal Testing . . . . .	52
b. Operating Deflection Shape Test . . . . .	52
3. Curvefitting . . . . .	53

a.	Coincident/Quadrature (Co/Quad) Fit . . .	53
b.	Circle Fit . . . . .	54
c.	Rational Fraction Polynomial Fit . . . .	54
G.	EMODAL-PC TEST PROCEDURE . . . . .	55
1.	Configuring the General Set-up . . . . .	55
2.	Geometry . . . . .	57
3.	Data Acquisition . . . . .	58
4.	Extract and Animate Shapes . . . . .	61
5.	Store Project . . . . .	63
6.	Recall Project . . . . .	64
7.	Prepare Reports . . . . .	64
IV.	CONCLUSIONS . . . . .	66
A.	ANALYTICAL MODELING . . . . .	66
B.	EXPERIMENTAL MODELING . . . . .	67
C.	COMBINED TESTING AND ANALYSIS . . . . .	68
V.	RECOMMENDATIONS . . . . .	69
A.	ANALYTICAL MODELING . . . . .	69
B.	EXPERIMENTAL MODELING . . . . .	70
C.	COMBINED TESTING AND ANALYSIS . . . . .	70
APPENDIX A	. . . . .	72
APPENDIX B	. . . . .	82

APPENDIX C . . . . .	90
APPENDIX D . . . . .	93
LIST OF REFERENCES . . . . .	98
BIBLIOGRAPHY . . . . .	100
INITIAL DISTRIBUTION LIST . . . . .	101



## LIST OF FIGURES

Figure 1	Sidewinder Missile . . . . .	2
Figure 2	AIM-9 Missile System . . . . .	3
Figure 3	Modeling Techniques . . . . .	6
Figure 4	PAL2 Program Execution . . . . .	13
Figure 5	MSC/PAL2 Suggested Order of Analysis . . . . .	21
Figure 6	MSC/PAL2 AIM-9 Geometry Model . . . . .	23
Figure 7	Combined Testing and Analysis . . . . .	33
Figure 8	Missile Modes for Single-Rail AIM-9 . . . . .	37
Figure 9	Spring - Mass Model . . . . .	38
Figure 10	Typical Shaker Setup . . . . .	43
Figure 11	AIM-9 Missile Modal Analysis Block Diagram . . . . .	47
Figure 12	AIM-9 Missile Modal Analysis Test Setup . . . . .	48
Figure 13	EMODAL-PC Test Procedure . . . . .	56
Figure 14	AIM-9 Missile Model Mode 1 Response . . . . .	84
Figure 15	AIM-9 Missile Model Mode 2 Response . . . . .	85
Figure 16	AIM-9 Missile Model Mode 3 Response . . . . .	86
Figure 17	AIM-9 Missile Model Mode 4 Response . . . . .	87
Figure 18	AIM-9 Missile Model Mode 5 Response . . . . .	88
Figure 19	AIM-9 Missile Model Mode 6 Response . . . . .	89

## ACKNOWLEDGEMENT

I would like to express my gratitude and appreciation to the following people, whose support and confidence in my abilities helped me make it through these last fifteen months.

First, I would like to thank Professor Ed Wu, my thesis advisor, for his guidance, patience and always optimistic outlook on life. It has been a privelege working with you, and an even greater pleasure knowing you.

Secondly, I would like to thank and acknowledge the support and advice I received from Mr. Ken Ranger of the MacNeal Schwendler Corporation, without whose help I might not have been able to get as far on the AIM-9 model as I did.

Next, I would like to thank Mr. Tim Twigg from the Naval Air Test Center, who not only got me the missiles in the first place, but also obtained much-needed information about the missiles on the spur of the moment.

My deepest gratitude goes to my wife Maureen, and our two children, Michael and Meagan. I can probably never make up for all those times when I couldn't go where you wanted to go, or do what you wanted to do, but please believe it was for a good cause. You may never realize how important your constant love, understanding and encouragement have been to me, not to mention the help with the typing. Thanks for being there when I needed you.

## I. INTRODUCTION

### A. BACKGROUND

Over the last 25-30 years, the P-3 Orion patrol airplane has performed its primary mission of Anti-Submarine Warfare (ASW) with virtually no defensive weaponry to protect itself on its solitary, long-range missions on the world's oceans. During its first 15-20 years of existence, the P-3 only carried armament in the form of mines and torpedoes, as well as "Zuni" rockets for ground suppression. It had been viewed primarily as a patrol airplane with minimal, if any, offensive capability. However, in the late 1970's and early 1980's, when the Harpoon anti-ship missile was integrated into the platform, the P-3 suddenly became an offensive weapon platform. Throughout all this time, it has remained highly vulnerable to, and defenseless against, any and all air threats.

The issue then becomes one of increasing the survivability of the P-3 against airborne threats. During the last few years, Defensive Air Combat Maneuvering (DACM) has been, and is currently being taught, to train P-3 aircrews in tactics that would lengthen the P-3's survival time in an air-to-air engagement. This training has shown promise, but it was soon realized that, if an air-to-air defensive missile were to be

integrated into the weapon inventory, the F-3 would have a much better chance of surviving, if not possibly destroying, the air threat.

This realization led to the F-3/AIM-9 proof-of-concept integration program, begun in 1989, and currently on-going at the Naval Air Test Center (NATC) at Naval Air Station Patuxent River, Maryland. In this effort, NATC was tasked to evaluate the separation characteristics of the AIM-9 Sidewinder missile, shown in Figures 1 and 2, as installed on F-3 outer wing station nine, and to define system requirements for possible use in a F-3. Wing station nine is of critical importance to the test program because it allows the carriage of an AIM-9 missile without losing a Harpoon wing station.

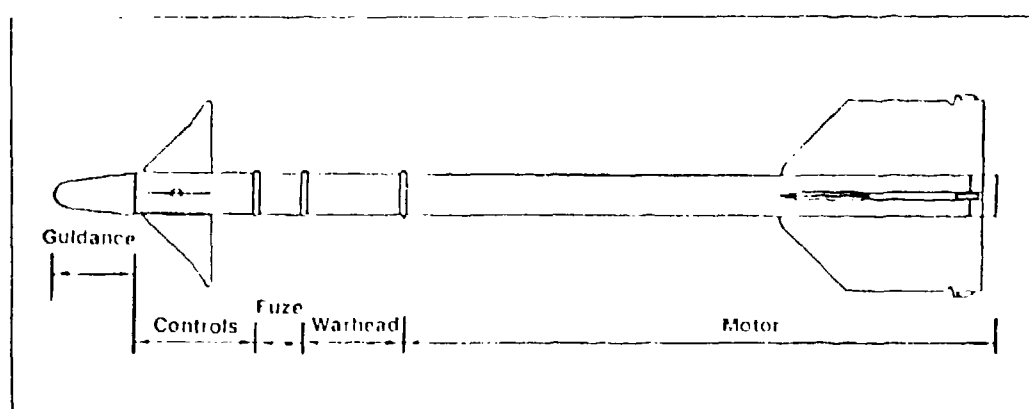


Figure 1 Sidewinder Missile

One of the main concerns with carrying AIM-9 missiles was whether or not the natural vibration modes and frequencies of the missile system would interfere with the two low frequency,

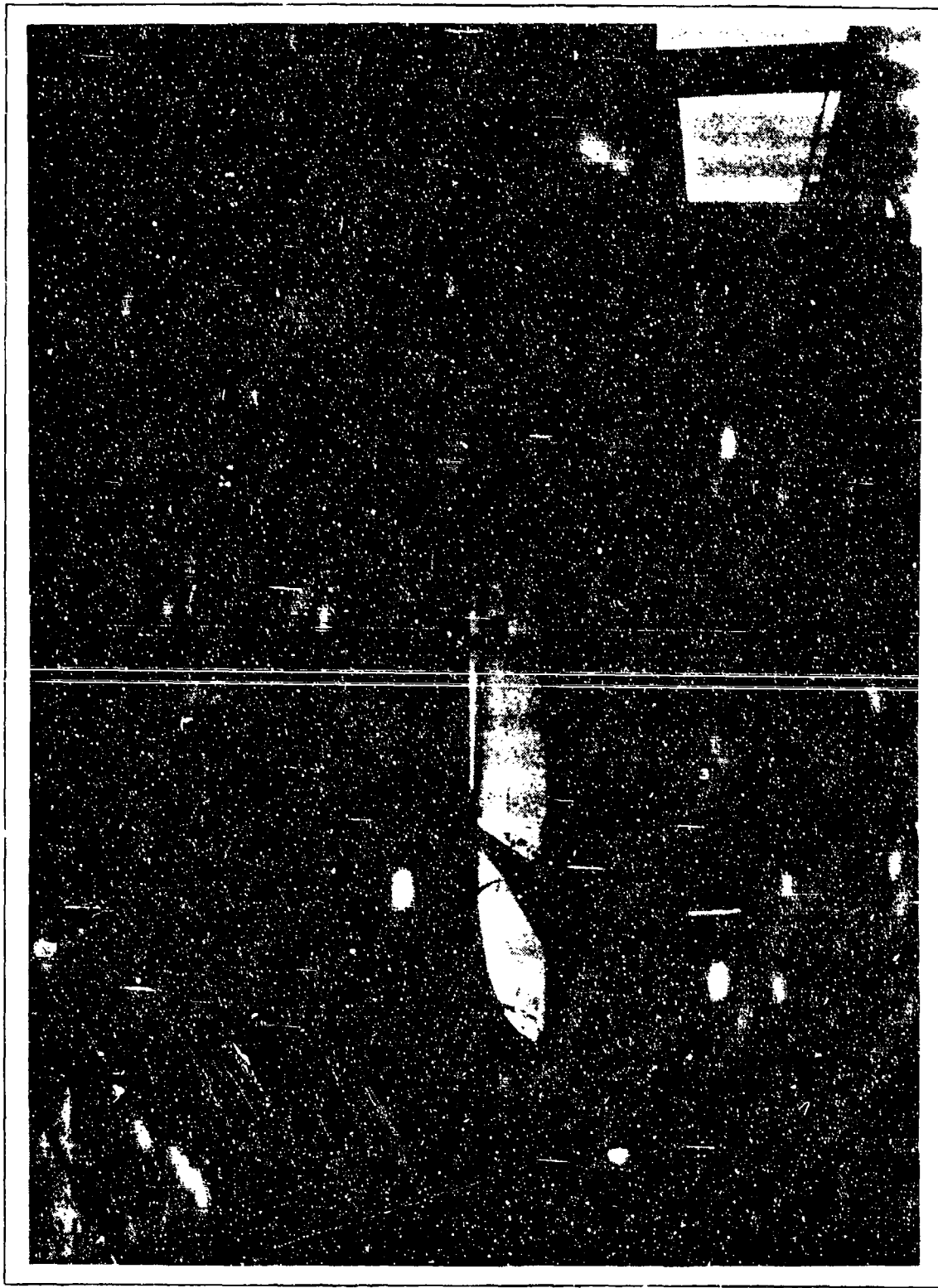


Figure 2 AIM-9 Missile System

outer wing vibration modes of the P-3: the outer wing bending mode at 4.7-8 hertz (hz) and the outer wing torsion mode at 17-22 hz. This could be determined by conducting a Ground Vibration Test (GVT), the results of which could then be incorporated into a computer model of the P-3 wing, leading to a determination of whether or not a potential flutter problem exists with an AIM-9 installation on a P-3 wing.

In support of this on-going program, Hollyer [Ref. 1] designed and constructed a GVT stand at the Naval Postgraduate School. A ground vibration characterization of the AIM-9 missile was conducted, resulting in the development of a two degree-of-freedom lumped-mass model, and experimental determination of the missile's resonance modes in pitch.

With the recent termination of the P-7A and P-3H programs, the P-3 community is now looking at the P-3 Orion II program to carry it into the 21<sup>st</sup> century. The Orion II will most likely have a beefed-up wing structure, necessitating an analysis of this wing in conjunction with the AIM-9 missile. This investigation responds to that requirement by concurrently developing a mathematical model of the AIM-9 missile using finite element methods to analytically determine its modal parameters, and setting up a modal test system to verify the mathematical model by experimentally determining the missile's natural frequencies, mode shapes and transient response. The analytical modeling, combined with the fully

instrumented experimental modal testing pursued in this investigation will lead to the following:

- Full characterization of the dynamic characteristics of the AIM-9 missile. When combined with an appropriate analytical model of the wing, these methodologies will allow the analytical assessment of different missile mounting locations and configurations without the risk and expense of flight tests.
- Modification of the parameters in the analytical AIM-9 model to analytically assess the changes in the modal characteristics for the upgrades and modifications of the missile. Upgrades and modifications may include propellant change in the rocket motor for range extension, and modified guidance and control system resulting in a mass change. Again, this is possible without the expenses of prototype construction and testing.
- Characterization of other candidate missile systems (e.g., AGM-65 Maverick and AGM-88 HARM) for the P-3 and its eventual successor.

#### B. MISSILE MODELING TECHNIQUES

To obtain a structure's vibration characteristics, two different methods can be used. The first method is analytical modeling which utilizes the finite element method, implemented in a computer software program. This is a "forward-type" of analysis, whereby the structure is partitioned into components whose mass, stiffness and damping are parameterized, and the structure's natural frequencies and mode shapes are predicted from these input parameters. The second method, experimental modeling, is usually referred to as modal analysis. This is an "inverse-type" of analysis, whereby the actual structure is subjected to a known forcing function, and the structure's

response to the excitation is measured at multiple locations. From the individual responses at different locations, the overall structure's stiffness and damping can be determined, along with the natural frequencies and modes of vibration. A block diagram of both of these methodologies is shown in Figure 3 [Ref. 2:p.1-7].

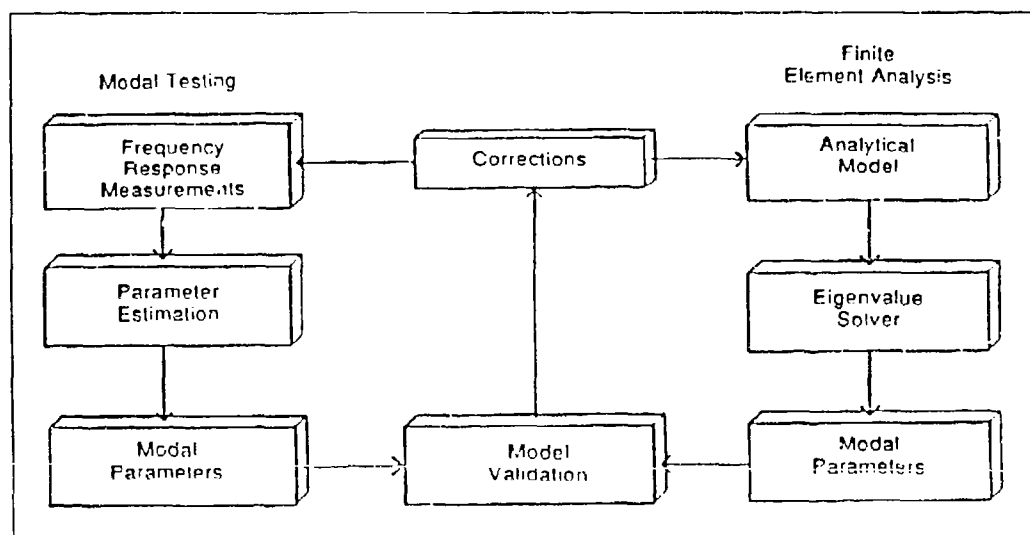


Figure 3 Modeling Techniques

### C. IMPORTANT DEFINITIONS

A compilation of terminologies is presented to facilitate the discussions of the finite element and modal analysis techniques. Definitions of the terms are available from textbooks of vibration analysis, and the following listing was adopted from two of these books [Refs. 3 and 4].

- Modal parameters - natural frequency, damping and mode shape



- Natural frequency - a frequency at which a system vibrates when disturbed from rest and then released; that frequency where, in theory, a minute force can produce large movements of the structure
- Resonance - occurs when a force coincides with the natural frequency; movement of the structure increases either until it is limited by damping or non-linear effects, or until something breaks
- Mode - resonance or peak in a Frequency Response Function
- Modal model - set of modal parameters representing the dynamic behavior of a structure
- Modal mass - mathematical concept that can be thought of as being proportional to the amount of mass actually in motion in a given mode shape; it is not the physical mass of the structure and cannot be measured
- Modal stiffness - mathematical concept that can be thought of as being proportional to the amount of stored energy in a structure in a given mode shape at the instant it reaches the point of extreme motion
- Damping - quantity that controls the sharpness of a resonance; usually expressed in percent of critical
- Frequency Response Function (FRF) - ratio of the response (either displacement, velocity or acceleration) at coordinate  $i$  to the excitation at coordinate  $j$  (for example,  $H_{ij} = X_i/F_j$ )
- Measurement - FRF acquired from a spectrum analyzer
- Degree-of-freedom (DOF) - measurement point and direction defined on a structure
- Driving point FRF - measurement made with the excitation and response at the same point and direction
- Reference transducer - transducer connected to the reference channel (channel A) of the analyzer; normally, this is the force transducer
- Response transducer - transducer connected to a response channel (channel B, C, or D) of the analyzer; normally, this is an accelerometer

- Mode shape - a set of shape coefficients at different locations along a structure that defines the structure's deformation at a specific frequency
- Real (normal) shape - purely real (or imaginary) numbers that describe the vibration pattern of the structure; all parts of the structure are moving either in phase, or  $180^\circ$  out of phase, with each other
- Complex shape - when each deformation value requires two numbers (magnitude and phase, or real and imaginary) to describe the vibration at a location; can have any phase relationship between different parts of the structure
- Curvefitting - matching a mathematical equation to a set of data points obtained from an FRF measurement

## II. ANALYTICAL MODELING

### A. FINITE ELEMENT METHOD BACKGROUND

In designing structures, the finite element method is often used to determine the structure's natural frequencies and mode shapes before it is actually built, in an effort to foresee any potential implementation problems which may occur. The finite element method can be viewed as a building block approach whereby the stiffness, damping, mass and material properties for the individual building blocks are known, or at least estimated, in advance and coded into one of the finite element analysis software programs currently on the market. In the initial design, the designer will have at least a general idea of the configuration of the structure, the location of components, and any mechanical interfaces and attachments. Constituent properties of the structure can be estimated, with iteration and confirmation of the changes leading to the final design of the structure.

The finite element method is not just a useful tool for designers. It can also be used to determine the natural frequencies and mode shapes of structures which are already built. Generally speaking, in the finite element method, structures are represented by discrete grid or node points which are connected by structural elements. In other words,

the finite element method can be viewed as a discrete representation of a continuous system made in order to simulate its structural behavior and response to expected forces or loadings. This representation is in the form of a mathematical model consisting of discrete elements connecting discrete nodal points. The more node points that are chosen to represent the structure, the more accurate the solution will be. Conversely, the fewer the number of node points chosen, the stiffer the model will be. If the aim of the designer or analyst is a detailed stress analysis of the structure, then the nodal density must be increased in regions of large stress gradients, such as around attachment points and around the location of applied forces. However, if the aim is a deflection shape analysis only, then fewer nodes may be used. [Ref. 5:Section 1, pp. 2-2, 2-3]

For either an existing structure or one that has yet to be built, a typical finite element analysis requires the following [Ref. 5:Section 1, p. 2-1]:

- Nodal point spatial locations
- Structural elements that connect the nodal points, representing the stiffness of the structure
- Mass properties of the structure
- Boundary conditions or structural constraints
- Static and dynamic load application

The mathematical model that results from the analysis can then be used to determine the structure's natural frequencies, mode shapes, stress levels and displacements. An added benefit of conducting a finite element analysis on a particular structure is that design iterations can easily be accommodated, cheaper and faster than by altering the actual hardware. For example, if the designer or analyst wished to determine the effect of additional mass on the structure's natural frequencies, all that would be required is modifying a few lines of software code, and rerunning the analysis. The effect of changing the material type, damping, shape or size of the structure can be handled easily as well. However, to achieve an accurate model, care must be taken in the assumptions made about the structure's geometry, material properties and so on.

## **B. MSC/PAL2 FINITE ELEMENT ANALYSIS SOFTWARE**

### **1. Introduction**

The finite element analysis software package used in this investigation was the MacNeal-Schwendler Corporation's MSC/PAL2 Advanced Stress and Vibration Analysis Software, Version 4.0. MSC/PAL2 uses the finite element method to solve for a structure's displacement, forces and/or stresses at different points on the body. The software has the capability of both static and dynamic analysis of a structure, although it limits the number of degrees of freedom, and hence the number of node points, for certain dynamic analysis

techniques. In addition to tabular output of nodal locations, element connectivity and nodal deformations, the software also has the capability to produce graphical output for analyzing model geometry, structural deformations, displacements as functions of time and frequency, animated deformation plots, and stress contour plots.

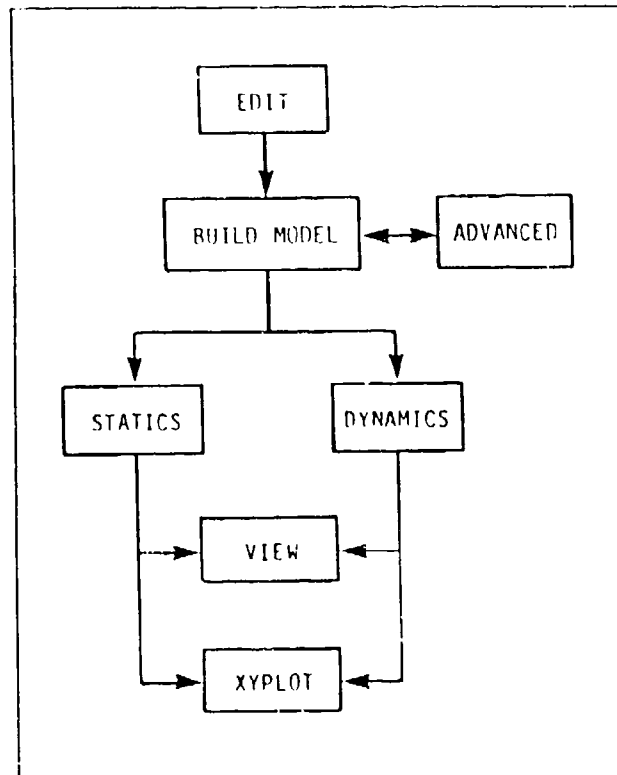
MSC/PAL2 requires an 80386 (or 80486) personal computer with at least a 20 megabyte (Mb) hard disk and at least 2 Mb of random access memory, with additional hard disk space being required for very large problems. Other requirements include at least one floppy disk drive, PC DOS 3.1 (or higher) operating system, an 80387 math coprocessor and either a CGA, EGA, or VGA monitor.

## **2. Basic Program Operation.**

MSC/PAL2 Version 4.0 is a menu driven program, beginning at the top level with six main menus. Each of these menus then branches out into varying numbers of submenus, depending on the application chosen. The interrelationship of these menus is shown in Figure 4, and is addressed in the discussion that follows.

### **a. File**

The file menu has only two choices. The first selection, "Configure", allows the user to change the disk drive destination to which the database files will be saved. It is primarily used to store databases on alternate drives



**Figure 4** PAL2 Program Execution

(for example A, B, D) for very large problems. The second selection, "Quit", is the command used to exit the program.

#### **b. Edit**

The Edit menu likewise has only two choices. The first choice, "Text file", allows the user to view, create or edit an existing text file utilizing the software's text editor. This is the option used to create and edit the model file and the load file(s) which are utilized in the analysis. The second choice, "MSC-MOD", is an interactive graphical preprocessor for the MSC/PAL2 program, enabling the user to

build and edit two-dimensional and three-dimensional finite element models representing actual structures. Instead of writing software code in a text file, MSC-MOD uses a menuing system overlaid on a graphical display which has the advantage of letting the user observe on the display the results of a particular command. MSC-MOD can be used in place of MSC/PAL2 to create a PAL2 model, or it can be used to edit an existing PAL2 model.

### ***c. Analysis***

The Analysis menu serves the purpose of providing commands for both building the model from the previously defined model text file, and also performing the actual static and/or dynamic analyses using the previously defined load file(s). The "Build Model" submenu generates a mathematical model of the structure defined in the model file, which consists of a set of second-order ordinary differential equations of motion, representing the mass and stiffness mathematical expressions for the structure. These generated system equations, when solved, give the structure's nodal displacements and element stresses. Since these equations are formed from the mass, stiffness and damping of each element making up the structure, and are specified for each nodal DOF in the system, matrices are employed to help keep track of all the different equations. The system stiffness matrix is built up from the stiffnesses of each element, which are dependent



on the size, shape, type and material make-up of each element. The element stiffnesses are defined via the PAL2 "BEAM TYPE" command, from which the beam cross-sectional area, bending moments of inertia, and torsional moment of inertia are computed. The system mass and damping matrices are computed by the program in an analogous manner, with the mass matrix originating from the mass density specification in the PAL2 "MATERIAL PROPERTIES" command, and the damping matrix originating from the PAL2 commands "DAMPER" and "DAMPING ELEMENT". [Ref. 5:Section 1, pp. 2-27 through 2-30]

The second submenu under the Analysis heading is the "Static Analysis" menu option. Static analysis is used to determine the stiffness and element stresses of the structure when subjected to some static load, which is defined in a static load file. The static loads are applied to the model (built using the "Build Model" submenu command) and the program computes the displacements. Additional loads can be applied to the same model, if desired, without building the model again. In fact, the analysis can be run repeatedly without rebuilding the model. After all of the loading cases of interest have been run, the "Data Recovery" option is invoked in order to obtain element stresses. [Ref. 6:Section 2, p. 3-8]

The third submenu under the Analysis heading is the "Dynamic Analysis" menu option. Dynamic analysis proceeds in a similar manner to static analysis, although multiple load

conditions cannot be run on the same model. The "Dynamic Analysis" menu option is very flexible in that it allows the user to compute a structure's natural frequencies and mode shapes (using the "Normal Modes Analysis" option), responses to frequency-dependent forces (using the "Frequency Response Analysis" option), and/or responses to time-varying loads (using the "Transient Response Analysis" option). Dynamic analysis of a structure should be run only after the static analysis, and the normal modes analysis should be the first dynamic analysis run to assess the overall dynamic characteristics of the structure.[Ref. 5:Section 1,p. 3-4]

#### **d. Graphics**

The Graphics menu of MSC/PAL2 has two submenus from which to choose: "View" and "XYPlot". The "View" option can be used in two ways. First, it can be used to check model geometry, prior to actually analyzing the model, to make certain that all node points are properly connected with elements. This is accomplished using the "F2: Get Dataset" option from the first-level "View" menu. "View" is also used after a model has been built to view the structural deformation to some load, including animation of the deformation. This is accomplished using the "F1: Get Model" option. The "View" option is used to view the graphical results of both static and dynamic analyses, in both the deformed and undeformed cases.

The second option under the Graphics heading is the "XYPlot" option. This option is primarily used to graphically represent frequency and transient response results in X-Y plot format, and provides graphical displays of response as a function of time, frequency, element number and subcase. Like the "View" option, "XYPlot" is menu-driven and uses the ten function keys to select the different plot options.

#### ***e. Composite Capabilities***

The Composite menu is a pre- and postprocessor to the Analysis section, used to create composite laminate descriptions for plate material properties and for printing or plotting composite results. Laminates with up to 200 layers made from up to ten different materials can be made. Since the AIM-9 missile is not of composite construction, this section of the MSC/PAL2 software was not used in this investigation. It is mentioned here simply for completeness. [Ref. 6:Section 3,p. 1-1]

#### ***f. Advanced Capabilities***

The Advanced menu presents some additional capabilities of the MSC/PAL2 software, and includes the following. The "Status" menu option shows whether the DOFs of the current model are active, eliminated or zeroed. The "Equations" menu option prints out the terms of the mass, stiffness and damping matrices for the most recent model created by the "Build Model" option. The "Expanded File" menu

option provides expanded MSC/PAL2 model data sets, which are significantly larger than the original data sets from which they were created, since all nodal points and element connectivity are explicitly specified. The "MSC/NASTRAN CONVERSION" menu option translates an MSC/PAL2 model dataset to MSC/NASTRAN format. Finally, the "REPLAY2" menu option, which is a separate program from MSC/PAL2, allows the user to continuously replay text, picture and animation files in a slide-show format. Since the MSC/PAL2 program was only used in this investigation to build a model file and obtain natural frequencies and mode shapes, these advanced capabilities were not utilized, and are mentioned here simply for completeness. [Ref. 6:Section 4,pp. 1-1 through 6-1]

### **3. MSC/PAL2 Analysis Procedure**

The analysis of a generic structure using the MSC/PAL2 finite element software program can be broken down into three general steps. The first step, system definition, is the process in which the actual structure is simulated by a mathematical model, defined by node points connected with elements. The analyst chooses a coordinate system, and the locations of all node points are specified relative to the origin of the coordinate system chosen. After all of the nodal point locations are defined using the "NODAL POINT LOCATIONS" commands, the structure's material properties (Young's modulus, shear modulus, mass density, Poisson's ratio

and tensile yield stress) are specified using the "MATERIAL PROPERTIES" command. Once the material properties have been specified, the element types are designated. Element type choices include several types of curved and straight beam elements, quadrilateral and triangular plate elements, and hexahedral, pentahedral or tetrahedral solid elements. Following the element type specification is the element connectivity process, using the "CONNECT", "DO CONNECT" and/or "GENERATE CONNECTS" commands to specify the nodal points that are joined by each element in the model. Finally, the boundary conditions (for example, attachment points) are specified using the "ZERO", "ELIMINATE", "ATTACH AT" and "OFFSET CONNECTION" commands. After the model has been fully defined, the "Build Model" menu option is engaged to assemble the system equations, at which time the model is ready for static and/or dynamic analysis.

The second general step, analysis, is the process in which different loading conditions are applied and the structure's static and/or dynamic behavior is computed. Static and dynamic loads are applied to the already-built model via separate load files, using the commands "DISPLACEMENTS APPLIED", "ACCELERATIONS APPLIED", and "FORCES AND MOMENTS APPLIED", among others, and the resulting nodal displacements and element stresses are computed. The analysis is conducted using the "Static Analysis" and "Dynamic Analysis" submenus, described previously.

The third general step, graphical postprocessing, is the process in which the static and dynamic results are presented in graphical format. The "Graphics/View" option can be used to show the structure's undeformed geometry, structural deformation and deformed animation. The "Graphics/XYPlot" option can be used to plot nodal displacements and accelerations, and also element stresses, as a function of time or frequency. These options were also previously described in detail.

For the conduct of an actual finite element analysis, these three general steps can be further broken down into a more specific plan of attack. This breakdown, shown in Figure 5, consists of a ten-step process. If the analyst is conducting a full-blown analysis, complete with frequencies, mode shapes, stresses and deflections caused by a variety of different loading conditions, then the process should be completed in its entirety, if possible. If, however, the analyst is interested in only certain parameters (for example, natural frequencies and mode shapes), then only those steps relevant to that particular analysis need be completed.

#### **4. MSC/PAL2 Limitations**

The maximum problem size for this version of PAL2 is dependent upon how many DOFs are in the model, as well as the type of DOF. Initially, there are  $G$  global DOFs in each model, where  $G$  is six times the number of nodal points, since

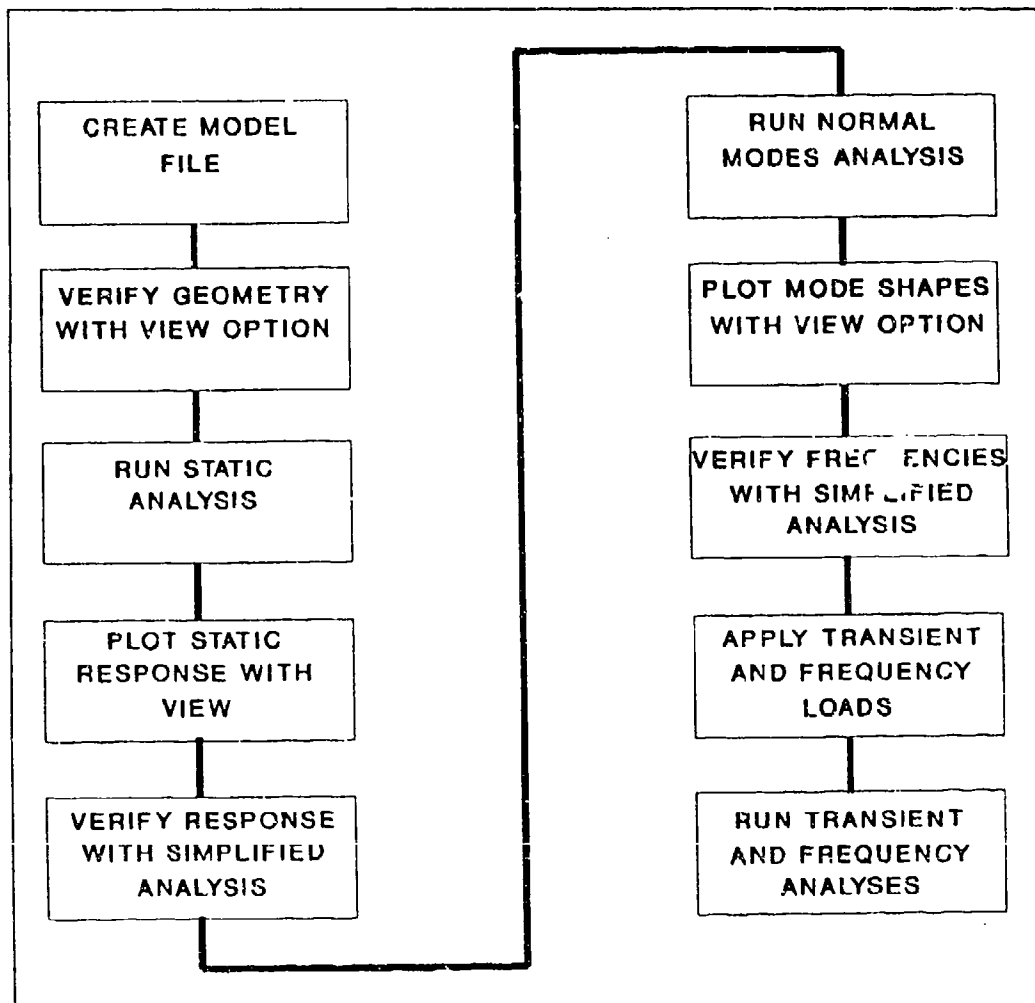


Figure 5 MSC/PAL2 Suggested Order of Analysis

each nodal point's motion can be represented by three translations and three rotations about the coordinate axis system chosen. The global DOFs are partitioned into the zeroed DOFs (created with the "Build Model" command "ZERO") and the free DOFs. The free DOFs are further partitioned into the eliminated DOFs (created with the "Build Model" command "Eliminate") and the active DOFs. With this in mind, the

problem size limitations for this version of the MSC/PAL2 software are as follows [Ref. 5:p. A-2]:

- Build Model: 2000 nodes, 12000 DOF
- Statics: 12000 free DOF or 225 active DOF
- Dynamics: Normal modes
  - Subspace iteration: 12000 active DOF
  - QR method: 225 active DOF
  - Jacobi method: 225 active DOF
  - Transient response: 225 active DOF
  - Frequency response: 150 active DOF

If a model has more active DOFs than shown here, and the analyst wishes to conduct a detailed analysis, then the number of active DOFs must be reduced via static condensation (for static analysis) or Guyan reduction (for dynamic analysis). MacNeal-Schwendler [Ref. 5:Section 1,p. 2-43] addresses in some detail the procedure for accomplishing this reduction. Since the main thrust of developing the AIM-9 model was to analytically determine the missile's natural frequencies and mode shapes, the normal modes subspace iteration method was used, and no reduction was necessary.

#### **C. MSC-PAL2 SIDEWINDER MISSILE ANALYTICAL MODEL**

##### **1. Model Geometry**

The MSC/PAL2 software code written to define the AIM-9 geometry is shown in Appendix A. The actual missile geometry



resulting from this code is shown in Figure 6, both with the hidden elements showing and with them not showing. From the code, it can be seen that a total of 706 node points were used to define this structure: 610 node points for the missile body, 36 for the front fins, and 60 for the rear wings. This large number of points was required to describe the missile

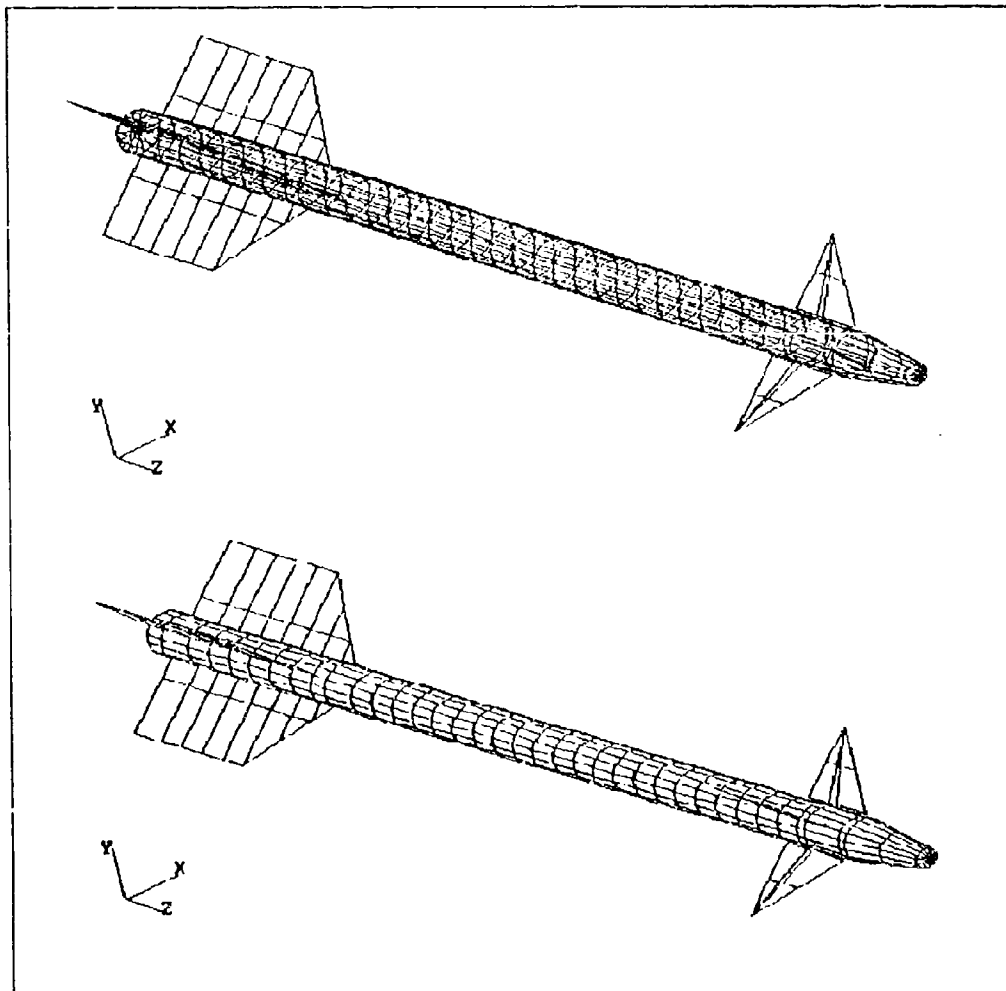


Figure 6 MSC/PAL2 AIM-9 Geometry Model

for several reasons. First, instead of using curved beam segments (which cannot be graphically displayed in the PAL2 software) to make up the cylindrical missile body, concentric rings of 16 nodes joined with small quadrilateral plate elements were spaced three inches apart to define the missile body. Secondly, these rings were spaced this closely because of the recommendation by MSC [Ref. 7] to avoid aspect ratios (length/width ratios) in the plate elements larger than four to one. Thirdly, a fairly uniform spatial distribution of node points is required to properly compute resonant frequencies and mode shapes. Since the missile model was built using the actual dimensions of the missile, the result was 35 concentric rings of node points making up the uniform cross-section of the missile body and three concentric rings of node points making up the tapered nose cone, all tapering into one node point representing the nose of the missile. Quadrilateral plate elements of 0.25 inch thickness were used to define the missile body and the tapered nosecone, while triangular plate elements of 0.25 inch thickness were used for the tip of the nose cone and the back plate. The three points where the AIM-9 missile attaches to the LAU-7 launcher rail were modeled by zeroing the displacements at the applicable node points. Translation in the x, y, and z directions and rotations about the y and z axes were zeroed for node points 85, 213, and 373. Rotation about the x axis was not zeroed,

since motion in the pitch direction is the primary motion of interest.

After the missile body was constructed, the front fins were added using a combination of quadrilateral and triangular plates, each of 0.10 inch thickness, with great care taken to keep the individual plate aspect ratios at less than four to one. Connecting the front fins to the missile body proved to be a problem because of the fact that the AIM-9's front fins are actually movable canards which exhibit rotation about their attachment points to provide missile guidance. The PAL2 software has no provisions for modeling this type of arrangement, so the fins were attached to the missile body via a small 0.35 inch plate, thereby prohibiting the rotation of the fins in the x-y plane about their attachment points. This resulted in a stiffness problem, with the fins rotating about the z-axis. Additional work is required in this area to reflect a more representative attachment method.

The rear wings were the next building block added to the model. These wings were modeled using a combination of 0.5 inch thick quadrilateral and triangular plates, and were rigidly attached to the missile body using previously defined missile body node points.

All that remained to complete the missile model was the addition of mass parameters to represent the motor, warhead, target detector and guidance and control components of the AIM-9 missile. This was accomplished using point

masses located at the midpoint of each component's length, with the exception of the motor component, which was represented by two point masses because of its 70 inch length.

Finally, the material composition of each element in the model was defined using generic steel values of Young's modulus, shear modulus, mass density, Poisson's ratio and tensile yield strength for the missile body and front fins, and generic aluminum values for the rear wings. These material properties, along with the nodal locations and element connectivity, are used to generate equations that, when solved, give the system displacements and stresses.

The completed AIM-9 missile model consisted of 706 nodal point locations, which resulted in a total of 4221 DOFs. The model was constructed of 717 elements of three different types: 660 quadrilateral plate elements, 52 triangular plate elements and five point mass elements. Quadrilateral plate elements were used as much as possible because they are less stiff than the triangular plate elements, thereby giving better and more accurate results [Ref. 5:Section 2,p. 2-3].

## **2. Determination of Modal Parameters**

Upon completion of the model geometry, and verification of the nodal point locations and connectivity using the "Graphics-View" option, the "Build Model" menu option was invoked to form the system stiffness, damping and mass matrices. These matrices arise from Newton's Second Law

of Motion, upon which all vibration work is based. Written as a second-order differential equation in the time domain, Newton's Second Law can be written as follows:

$$M\ddot{x}(t) + C\dot{x}(t) + Kx(t) = f(t) \quad (1)$$

where  $M$  = system mass matrix

$C$  = system damping matrix

$K$  = system stiffness matrix

$x(t)$  = nodal displacement vector

$\dot{x}(t)$  = nodal velocity vector

$\ddot{x}(t)$  = nodal acceleration vector

$f(t)$  = applied force vector

After the model was built, a normal modes analysis was conducted to determine the missile's natural frequencies and mode shapes. The system equations formed for a normal modes analysis include the system stiffness and mass matrices, written in matrix notation as follows:

$$M\ddot{x} + Kx = 0 \quad (2)$$

The right-hand side of this equation is set equal to zero since there is no forcing in normal modes analysis. All motion is assumed to be steady-state harmonic motion. The structure, in this case the AIM-9 missile, vibrates at some frequency,  $f$ , with each point moving either in phase with the other points or out of phase. Therefore, the nodal acceleration vector may be written in terms of the displacement vector as follows:

$$\ddot{x} = -(2\pi f)^2 x \quad (3)$$

Substituting this into Equation 2 gives:

$$[K - (2\pi f)^2 M]x = 0 \quad (4)$$

In the case of the AIM-9 missile, the dimensions of the mass and stiffness matrices is 4221 x 4221, since the model has 4221 active DOFs. One can immediately see the necessity of using a computer program for this type of analysis, as the manual manipulation of this many equations would be virtually impossible. The nontrivial solution to Equation 4 yields the system resonant frequencies, each of which has associated with it a characteristic vibration, or mode shape. Since there is no forcing in a normal modes analysis, the scaling values for each mode shape are arbitrary, and only the relative shape is important. [Ref. 5:Section 1,p. 2-40]

A normal modes analysis can be accomplished by three different methods. The default method of eigenvalue extraction, the QR method, is much faster than the Jacobi method, which is a more accurate method for unrestrained structures. Both of these methods require the system equations to be reduced in size to the point where there are no more than 225 active DOFs. This reduction is accomplished with the PAL2 commands "Eliminate" and "Activate". Since typically only the lowest few modes are excited to any appreciable degree by dynamic forces, these are the modes of primary interest to the designer or analyst. The subspace iteration method is the method of choice when only the lowest frequencies are needed, because it accomodates up to 12000

active DOFs and no reduction of the system equations is required. This method solves for the eigenvalues and eigenvectors with the mass and stiffness matrices in the form of Equation 4, obtaining these equations from the physical properties of the structure coded into the model, and was the method used for conducting a normal modes analysis of the AIM-9 missile model. The results of this analysis are given in Appendix B.

### **3. Modification of Modal Parameters**

The modal parameters of the AIM-9 model, specifically the natural frequencies and mode shapes, may be readily modified to reflect alternative arrangements of the missile configuration, simply by altering the mass, stiffness or damping properties input into the model. In the construction of the missile model, various different configurations were analyzed to determine the effect of changing these properties. The results of the five cases analyzed are presented and discussed in Appendix C. It can be seen from these results that the addition or movement of the point masses can have a substantial effect on both the natural frequencies and the mode shapes. Adding mass to the structure shifts the resonance to a lower frequency, as does decreasing the stiffness, while removing mass shifts the resonance to a higher frequency, as does increasing the stiffness. This is helpful in avoiding a known excitation. Moving the location

of a point mass only a short distance had the effect of completely changing the mode shapes. Changing the material properties of parts of the structure also affected the response. Since these results were obtained from a normal modes analysis in which only the relative shape of the mode is important and not the amplitude, damping is not addressed. However, damping would need to be increased to reduce the response at a particular resonance.

Knowing the effects of changing the various properties of a model enables the designer or analyst to investigate different design iterations or feasibility studies prior to making any hardware changes. For example, if ascertaining the effect of adding more fuel to the AIM-9 missile to extend its range was desired, mass would be added to the existing model to represent the increased fuel required, and additional node points would be added to represent the increased length of the fuel tank component. A new normal modes analysis could then be run, and the new natural frequencies and mode shapes obtained. Another possible missile upgrade includes incorporating a new type of seeker head. The effects of this potential modification could also be observed quite easily. For these studies to be of value, it is imperative that the baseline model accurately represent the actual structure not just geometrically, but in the modal responses as well. For this reason, it is important to utilize the combined testing and analysis approach to first obtain the missile's modal

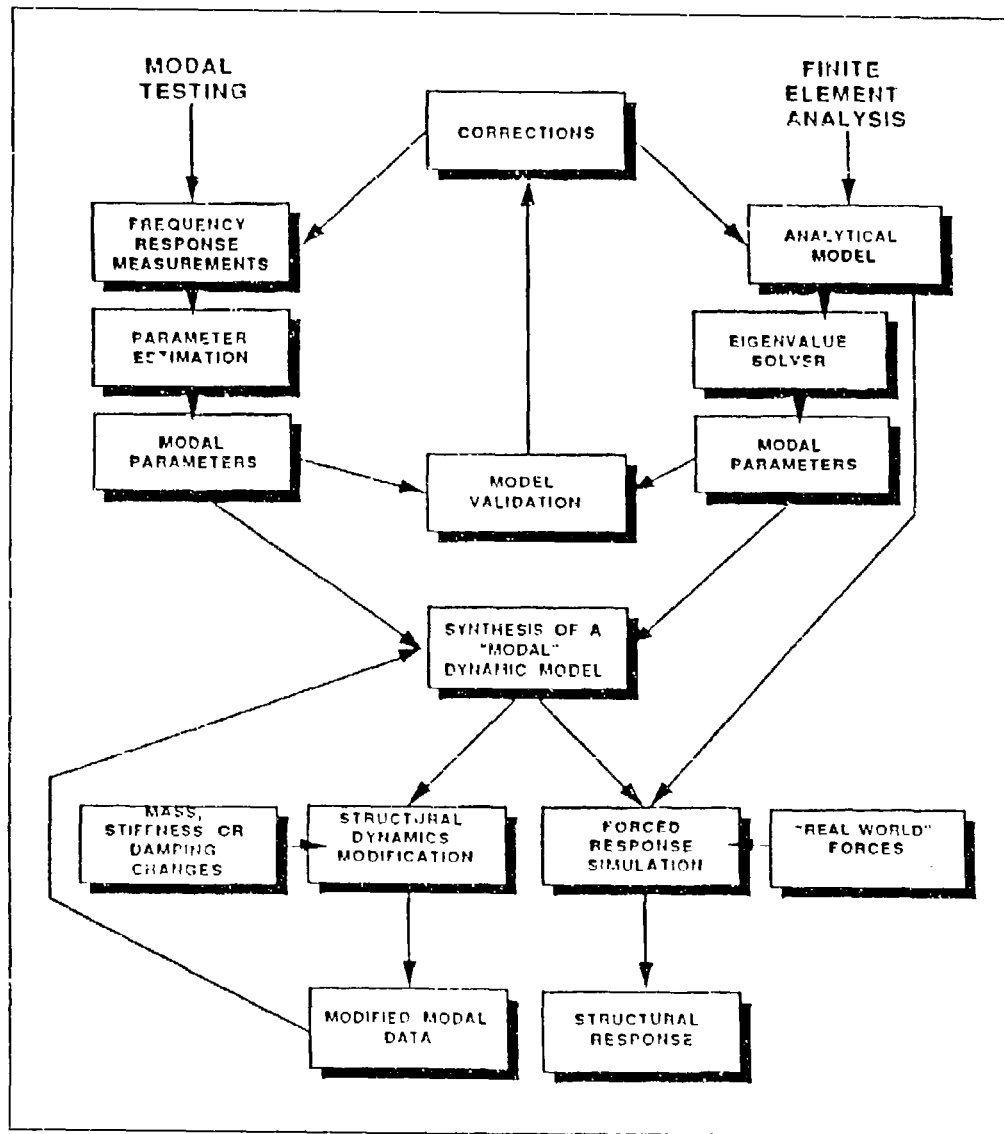


parameters experimentally, then modify the analytical model, to match the experimental results. It is at this point that the design iterations or feasibility studies can be conducted on the analytical model with a reasonable amount of confidence in the results obtained.

### III. EXPERIMENTAL MODAL ANALYSIS

#### A. GENERAL

In the same vein that the finite element method can be considered a "forward-type" analysis, in that a structure's constituent properties are input and the response is predicted, modal analysis can be considered an "inverse-type" analysis, in that a structure's response to an excitation is measured, and from the resulting transfer function, the structure's modal parameters can be obtained using analytical parameter estimation methods. The major advantage of conducting a modal analysis of a structure is that simplified mass-spring-damper models can be constructed with a good degree of certainty from the actual measured response of the structure. The drawback of modal testing is that design iterations or feasibility studies cannot be accomplished unless the structure is physically modified. The best approach is to combine the two methodologies, by conducting a modal analysis of the test structure to obtain the actual response, and then use this information to make the PAL2 model more accurate for use in future design iterations. This combined testing and analysis approach is shown in block diagram form in Figure 7 [Ref. 2:p. 3-4].



**Figure 7** Combined Testing and Analysis

Structural Measurement Systems, Inc. [Ref. 2:p. 1-2] defines modal analysis as the process of characterizing the dynamic properties of an elastic structure in terms of its modes of vibration. In the majority of present day modal

analysis practice, as well as in finite element analysis as mentioned previously, the motion of a physical system can be assumed to be adequately described by a set of simultaneous second-order linear differential equations of the form:

$$M\ddot{x}(t) + C\dot{x}(t) + Kx(t) = f(t) \quad (5)$$

If a system has  $n$ -DOFs, then the above vectors are  $n$ -dimensional and the matrices are  $n \times n$ . These system equations are transformed into their equivalent transfer function form by taking the Laplace transform, resulting in the relation:

$$X(s) = H(s) F(s) \quad (6)$$

where  $H(s)$  is called the transfer matrix, which can then be written in terms of modal frequency, damping and modal vectors. It is important to note that each row and column of the transfer function matrix contains the same modal vector, multiplied by some component of itself. Therefore, only one row or column of the transfer matrix needs to be measured in order to identify all the modal parameters of a structure, provided the following assumptions are met. The first assumption is that the motion of the structure can be described by linear second-order equations. The second assumption is that symmetry of motion exists, described by Maxwell's Reciprocity Theorem, which implies that the FRF measured between any two DOFs is independent of which of them is used for excitation or response. The third and final assumption is that no more than one mode exists at each pole location of the system transfer matrix. [Ref. 8]

Modal analysis is conducted on a structure for various reasons. Perhaps one of the most important is to verify and help improve the accuracy of finite element models. Experimental measurements obtained on the structure being analyzed result in a physical check of the accuracy of the mathematical model. If the model predicts the same modes and frequencies as those obtained experimentally, then the model is considered to be accurate. Otherwise, the model must be modified to match the same response as that obtained experimentally, assuming the experimental response is correct. Other important reasons for doing modal testing and analysis include troubleshooting noise and vibration problems, quickly evaluating fixes made to structures in order to solve noise and vibration problems, and formulating dynamic models for parts of structures that are too difficult or time-consuming to model analytically. In this investigation, a modal analysis test setup was completed to be used to analyze the response of the structure of interest (AIM-9 missile) in conjunction with another structural member (P-3 outer wing).

## **B. PREVIOUS WORK ACCOMPLISHED**

### **1. Test Setup**

Work was begun on this effort by Hollyer [Ref. 1] who designed and built the GVT stand, which was then used to conduct a vibration characterization of the AIM-9 missile (in pitch only), employing forced oscillatory inputs over a

frequency range of 0-50 hz. In this test, Hollyer used analog instrumentation (vice a digital spectrum analyzer) in which a shaker assembly was utilized in conjunction with a function generator to excite the structure sinusoidally. To measure the missile's response, two piezoelectric accelerometers were used, one mounted rigidly in series with the shaker assembly (the reference transducer) and one roving over the missile to read magnitude and phase shift at various locations on the missile (the response transducer) using an oscilloscope.

## **2. Previous Results**

Using the above apparatus, the missile system was excited in pitch, with the excitation force varying from 10 to 40 pounds peak-to-peak. Pitch modes were observed over a band of 24.4 to 25.6 hz and near 34 hz, and a heave mode was observed near 38 hz. The first elastic mode of the missile, obtained by placing the shaker at the node of the first rigid mode, appeared to occur at about 52.5 hz. However, this mode was terribly polluted with the motion of the GVT stand itself. Exact frequencies could not be determined due to the inaccuracies of the data collection equipment used. The results from this test are shown in Figure 8. Since the primary thrust of this effort was to determine the natural frequencies and mode shapes of the missile in pitch, the GVT was built and stiffened for excitation in the longitudinal direction only. Even so, the system was excited in sway and

yaw, with a sway mode observed at about 7.2 hz and a yaw mode at about 9.8 hz. However, these modes are very suspect due to significant motion of the GVT during excitation. To obtain more accurate missile response in sway and yaw, the GVT would need to be stiffened laterally.[Ref. 1]

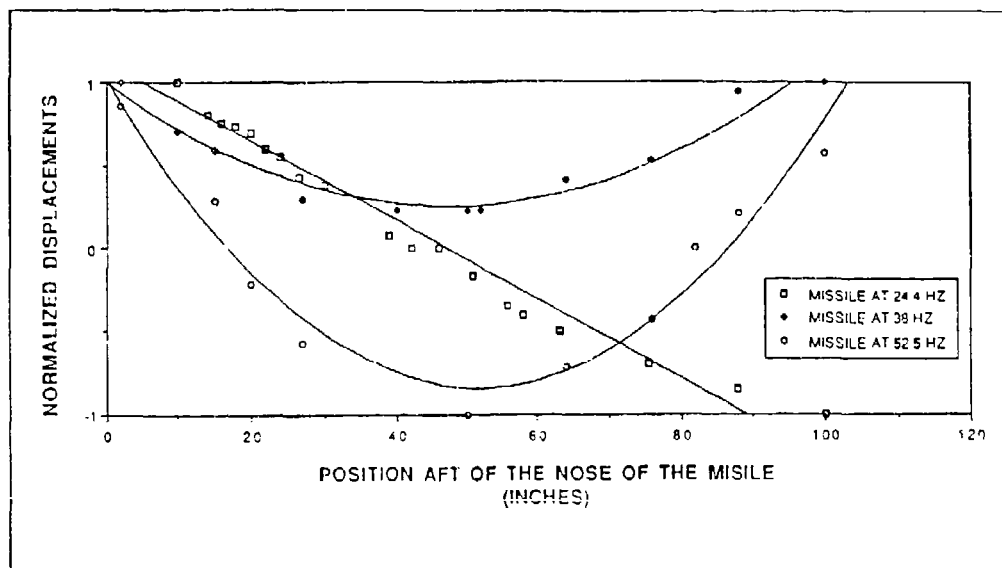


Figure 8 Missile Modes for Single-Rail AIM-9 [Ref. 1]

After the above modes were obtained, the missile system was modeled as the simplified spring lumped mass model shown in Figure 9. This model was represented as a two lumped mass system possessing no internal damping, with two springs to provide the two degrees of freedom to model the lowest resonance of the pitch and heave modes. The theory, logic and thought process behind this model can be found in more detail in Hollyer's investigation [Ref. 1].

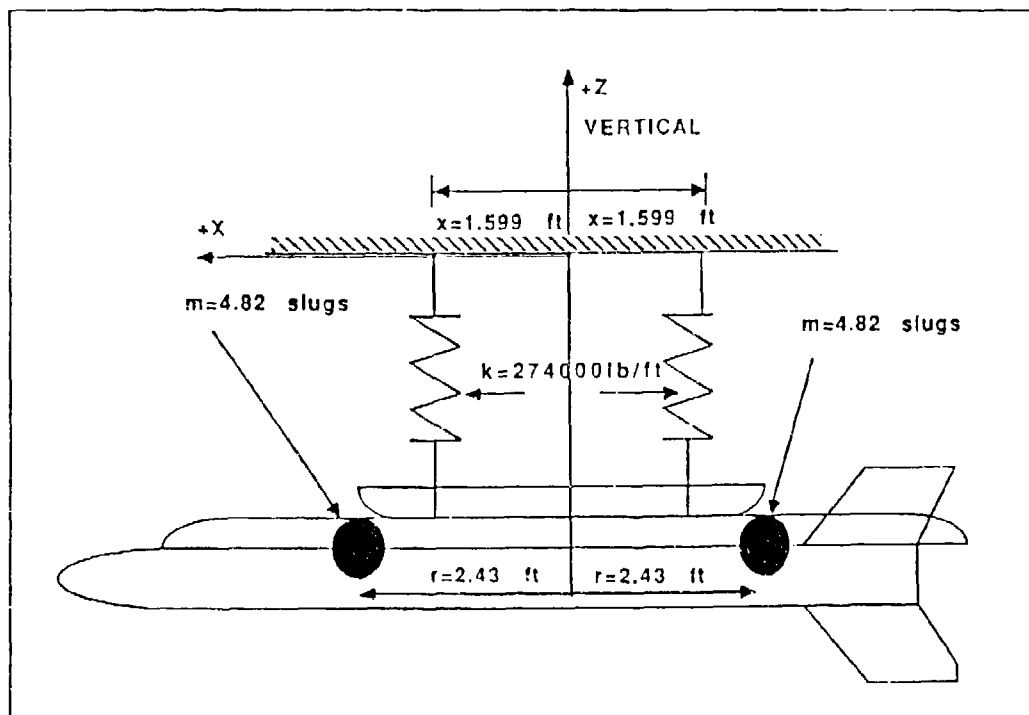


Figure 9 Spring - Mass Model [Ref. 1]

### C. MODAL TEST SETUP CONSIDERATIONS

#### 1. Transducer Selection

Prior to conducting a modal test, a multitude of considerations must be taken into account. Perhaps one of the most important considerations is the selection of a transducer. A transducer is a device which converts the vibratory motion of the structure into an analogous electrical signal. Transducers come in three basic types: displacement transducers, velocity transducers and accelerometers. The accelerometer is by far the most common and is replacing the velocity transducer in the majority of today's applications.



because of its superior phase and frequency response. One of the most common accelerometers in use today is the piezoelectric accelerometer, which is a small, lightweight transducer, useable over a wide frequency range. Since the output of a piezoelectric accelerometer is a very low voltage, a charge amplifier must be used to amplify the signal to a high enough level to be displayed by the spectrum analyzer. A recent development in accelerometers is the integrated-circuit-piezoelectric (ICP) concept, in which amplifiers in the form of microelectronic integrated circuits are built into the transducer. This precludes the need for a charge amplifier, but a constant current power supply is required to operate an ICP transducer.

Considerations in selecting an accelerometer for a specific project include sensitivity, frequency range, size, and long-term stability [Ref. 9]. If one or more of these properties is important for a particular application, tradeoffs may have to be made, and some of the other properties may have to be sacrificed. Accelerometers are usually selected to have a sufficiently high resonance frequency so that the frequency response can be considered flat throughout the operating frequency range. Accordingly, the accelerometer reference frequency should be at least five times the maximum frequency of interest [Ref. 10]. By following this rule, slight variations in resonance frequency due to different mounting methods can be ignored.

## 2. Transducer Mounting

The primary consideration in selecting a transducer mounting method is that the mounting method does not introduce any impedance into the system that would result in errors in the generated data. Various mounting methods are available, including stud mounting, wax mounting, cementing and hand-held probes and magnets. The preferred method is to use an isolated stud mount whereby the transducer is physically attached to the structure via a screw stud. This method provides the best frequency range, accuracy and long-term stability, and problems with dust, moisture or temperature are normally not encountered. However, this is a somewhat permanent method which modifies the test structure. If the test structure is one in which drilled and tapped holes are not desired (for example, a missile), the next best alternative is beeswax. This method allows for quick setup and transducer movement, while still providing for a moderate frequency range and accuracy. Manufacturer testing [Ref. 11] has shown that frequency response equivalent to stud mounting can be achieved with a thin coating of wax on clean, flat surfaces. If the surface is curved, a composite mounting base can be used, which can easily be sanded for mounting to contoured surfaces. Cementing is sometimes difficult because of the setting time required, and also because of adhesion problems to certain surfaces. Magnets allow for quick setup and movement, but this method is frequency-limited due to the

lack of a rigid attachment. Also, mass loading of small structures may occur. Hand-held probes, while being the simplest method, are not recommended.

### **3. Transducer Calibration**

It is not necessary to calibrate the transducers used in a modal test if all that is desired is the acquiring of a structure's mode shapes. However, transducer calibration is essential when developing a modal model to be used for structural modification studies. There are basically three types of transducer calibration that can be used. Absolute calibration of a single transducer requires a known source (for example, 1g at 1Khz) which is fed through the transducer and read directly on a spectrum analyzer. Referential calibration requires a well-calibrated transducer to be used as a standard to which the user's transducers are referenced. Finally, the constitutive or ratio calibration, which combines aspects of the absolute and referential methods, uses some constitutive law (for example, dividing all measurements by the ratio  $(A_{volt.}/F_{volt.}) * m$ ) to relate a pair of transducers to each other.[Ref. 2:p. 10-2]

### **4. Excitation Techniques**

#### **a. Shaker Excitation**

The objective of an exciter in a modal test is to input a force into the structure under test to excite all the modes of interest. Various excitation techniques have been

developed to help the analyst solve a particular vibration problem. One of the most common techniques is to use a hydraulic or electro-mechanical shaker assembly. A typical shaker setup for a modal test is shown in Figure 10. The shaker should be attached to the test structure in such a way that excitation of the structure occurs at a single point and in a single direction. A thin flexible "stinger" with a force transducer mounted at the end can be used to reduce bending moments. Using a shaker assembly requires a certain amount of fixturing, but it gives good control of the excitation bandwidth and level, and allows several choices of excitation signals. These signals are of two primary types: deterministic excitation (which includes sinusoidal sweep, periodic chirp, impulse and step relaxation) and random excitation (which includes pure random, pseudo random, periodic random and burst random). Each of these types of excitation techniques has its advantages and disadvantages. These characteristics will not be listed here, but are discussed in great detail by SMS, Inc. [Ref. 2:pp. 9-2 through 9-44]. The point is that the analyst must select an excitation technique based on the considerations and constraints of a particular application. Examples of these considerations are test measurement time, signal leakage control, bandwidth control, level of signal-to-noise ratio and distortion removal.

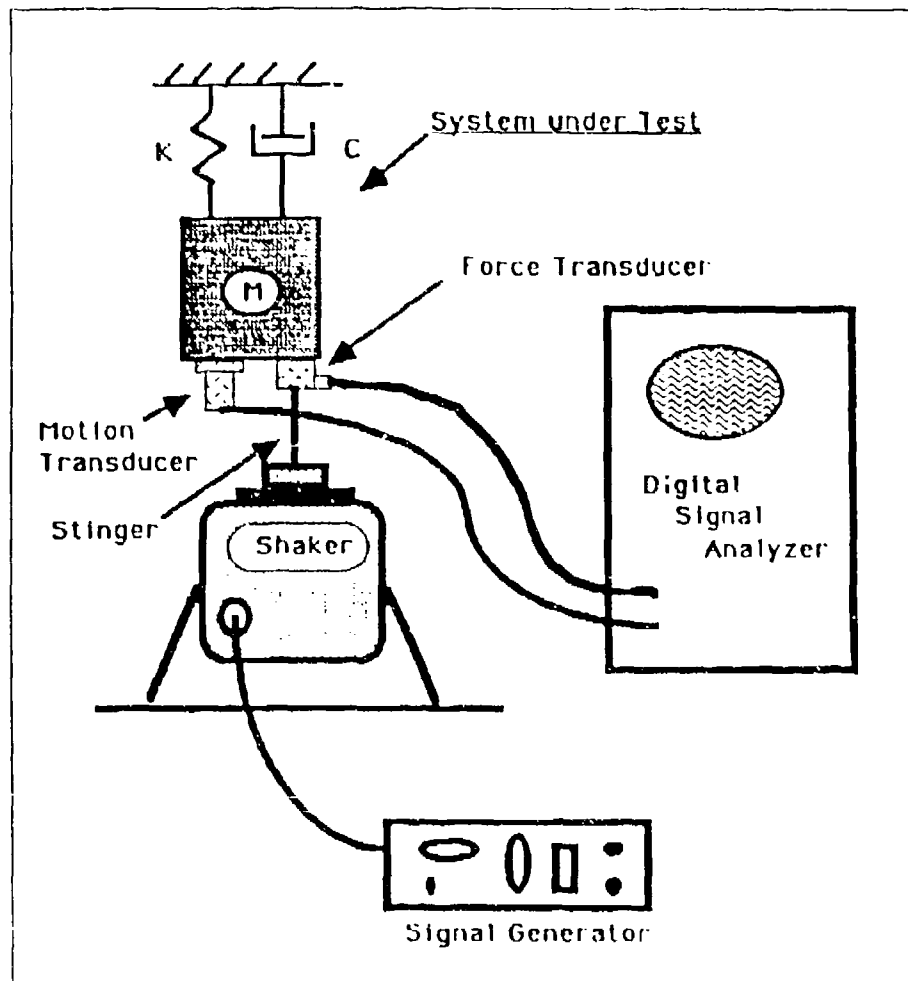


Figure 10 Typical Shaker Setup

#### *b. Impulse (Impact) Excitation*

The latest advancement in modal analysis is the use of an impulse, or tap, hammer to impart broadband excitation to the test structure, thus exciting all modes simultaneously. The excitation force response can be altered by changing the hammer tip material, which is determined by the frequency range of interest. A soft rubber tip will result in a longer

impact time which will concentrate the input energy into the lower frequency ranges. Conversely, a hard steel tip will result in shorter impact times for cases where a higher frequency range is of interest. The impact technique has several advantages over a shaker assembly in that it is fast, easy, low cost, convenient and portable, and it does not require any external signal source. However, it provides only a low input energy with no randomization of the input, a low signal-to-noise ratio, and no control of the frequency content if a particular excitation bandwidth is desired. Corelli and Brown [Ref. 12] state that, in general, impact excitation is the worst possible excitation method to use for most structural tests, as the technical disadvantages far outweigh the technical advantages. However, the convenience and quickness of impact testing is so dominant that it generally outweighs the technical disadvantages and is often chosen as the excitation technique.

#### **5. Specifying the Number of Degrees-of-Freedom**

As stated earlier, a DOF is a measurement point and direction defined on a structure. Any free point on a structure generally has six DOFs: three translational and three rotational. When deciding on the number of DOFs to use for a particular test, the analyst must consider the purpose of the test, and the geometry of the actual structure under test. If the purpose of the modal test is simply to verify

modal frequencies previously predicted analytically, then only a few DOFs would be required. However, if the purpose of the test is to construct a mathematical model interpolated from modal measurements at discrete locations, then sufficient DOFs must be used, taking the structure's geometry into account, to ensure that all mode shapes in the frequency range of interest are distinguishable and linearly independent. It is the geometrical complexity of a structure's mode shapes which ultimately determines the number of DOFs required. Known mode shapes of similar structures tested previously should be used whenever possible to determine how many DOFs with which to begin. Care should be taken in not using more DOFs than are necessary to define the mode shapes for a structure, as this results in longer computation and test time.

#### **D    SIDEWINDER MISSILE MODAL ANALYSIS TEST SETUP**

##### **1.   Test Structure**

The GVT stand utilized in the missile modal analysis setup is located on the basement floor of Malligan Hall at the Naval Postgraduate School in Monterey, California. The GVT is the same structure constructed and used by Hollyer [Ref. 1], except for the addition of a dozen additional fasteners joining the top and bottom plates to the 8 inch by 8 inch I beams, and two 14 inch bolts joining the top and bottom plates together for the purpose of providing further stiffening of the structure. This structure was initially designed to

provide minimal motion in the frequency range of interest (0 to 50 hz) to longitudinal test member excitation, with an initial resonant frequency of approximately 52.5 hz. The structure has no added stiffening for excitation in the lateral direction.

## **2. Test Equipment**

A basic modal test system is usually comprised of a spectrum analyzer, a computer with associated modal analysis software, a shaker or impact hammer to provide the excitation (with an external function generator, if needed), a force transducer to provide driving point measurements, response transducers to measure the response to the excitation, and the actual structure to be tested. The Sidewinder missile modal analysis test setup is depicted in block diagram form in Figure 11, and the actual setup is shown in Figure 12. Specific details of the individual components of this test setup are discussed in Appendix D.

The spectrum analyzer used in this test setup is the Scientific-Atlanta Model SD-380, with the four channel option installed. This option allows the parallel sampling of four channels of simultaneous information, thus permitting the force transducer to be connected to channel A, and the three Flexcel response transducers to be connected to channels B, C, and D. The SD-380 used in the missile test setup also has a Synchronous Signal Generator (SSG) capability, precluding the



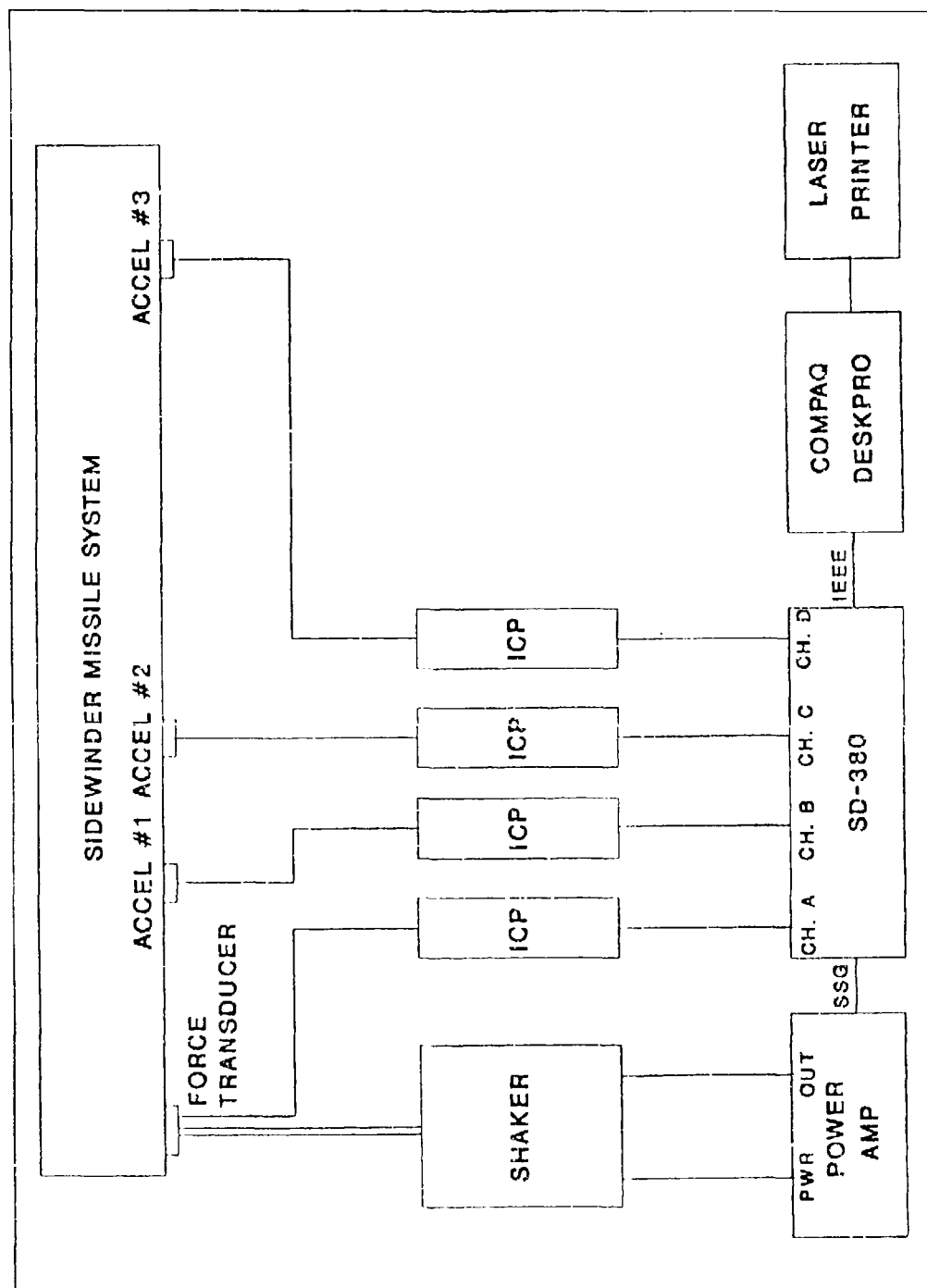


Figure 11 AIM-9 Missile Modal Analysis Block Diagram

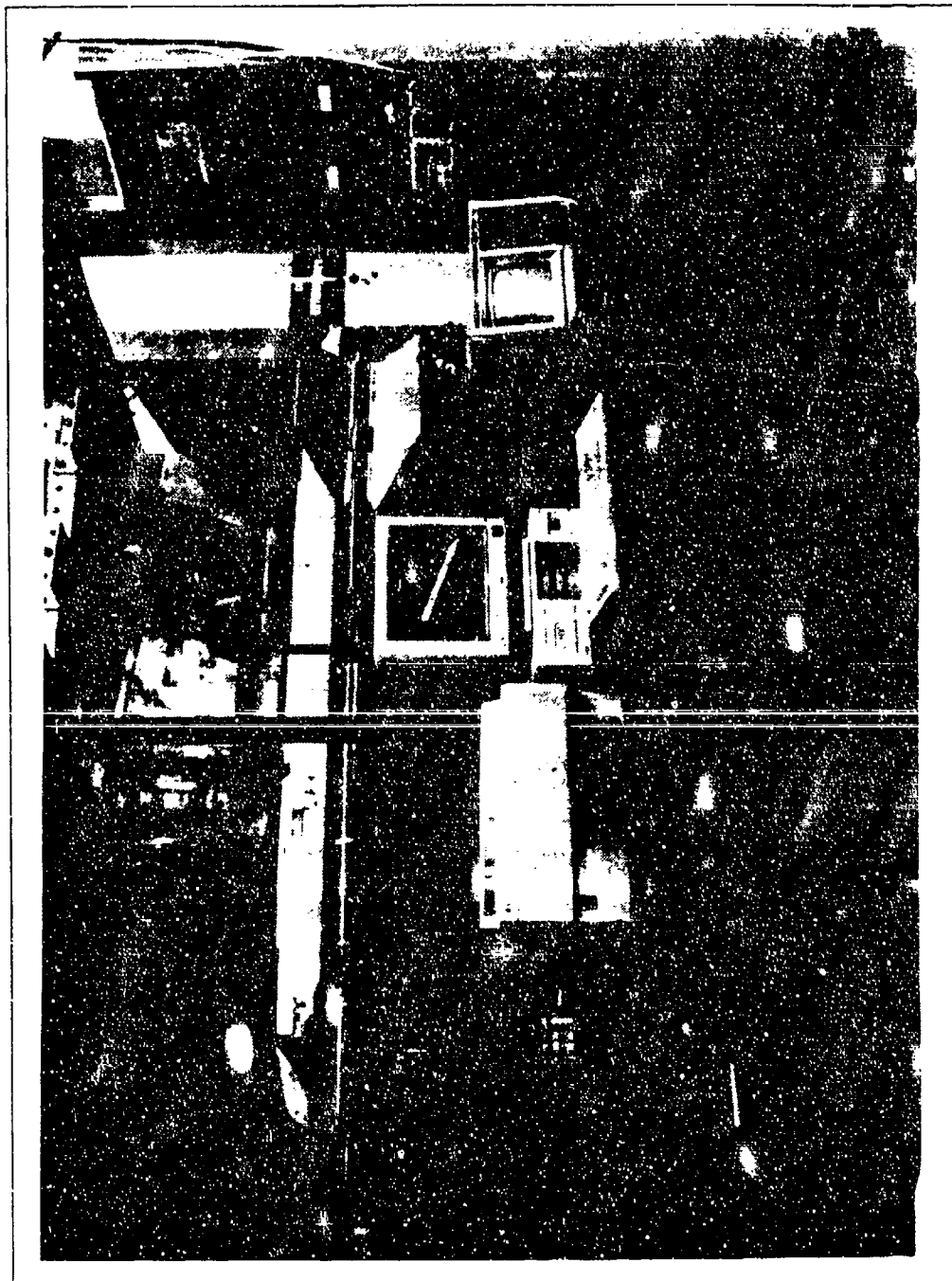


Figure 12 AIM-9 Missile Modal Analysis Test Setup

need for a separate function generator to send the excitation signal to the shaker. The SSG signal output types available include broadband white noise, pseudo random noise, band limited noise, sine and burst random. The SD-380 communicates with the computer, specifically the EMODAL modal analysis software package, via an IEEE-488 interface, using a General Purpose Interface Bus (GPIB), specifically the National Instruments AT-GPIB interface board. This interface allows the transfer of measurement data from the spectrum analyzer to the EMODAL program, or from the analyzer to an optional digital pen plotter.

In an actual modal test, the SD-380 is used to analyze a vibration signal, in this case acceleration, for its frequency content. When a structure is excited by white noise (full spectrum excitation), the response is the superposition of all the modes of the structure. The analyzer determines FRFs from the different measurement locations on the structure, and performs a Fast Fourier Transform to break out the separate modes at each location on the structure. The FRFs are shown as a series of peaks, with characteristic center frequencies, identifying them as resonances. It is these FRFs that are transferred to the EMODAL-PC software, which creates a mathematical expression for each FRF, and extracts the mode shapes and modal parameters. The software can then calculate and display a modal animation for all resonances of interest.

## **E. MODAL-PC SOFTWARE BACKGROUND**

### **1. Introduction**

With the growth in popularity of modal analysis in recent years, several good modal analysis software packages have recently been released. One of the packages currently on the market is EMODAL-PC by Entek Scientific Corporation. EMODAL-PC is a user-friendly personal computer-based software package, specifically designed and written for modal testing, operating deflection shape analysis and vibration troubleshooting. It allows 250 measurement locations for real shapes, 125 measurement locations for complex shapes, 750 total DOFs and 20 mode shapes.

EMODAL-PC requires an MS-DOS-based PC/XT/AT, 80386, or 100% compatible computer, with at least a 10 Mb hard disk and at least 640 kb of random access memory (of which at least 420 kb must be available for use by EMODAL-PC). Other requirements include at least one floppy disk drive, an 8087, 80287 or 80387 math coprocessor, a CGA, EGA or VGA monitor, and an IEEE-488 interface port on the computer on which it is to be used.

### **2. Major Features**

The version of the EMODAL-PC software used in the Sidewinder missile modal analysis test setup is version 2.75. The major features of this version of the EMODAL-PC software include the following [Ref. 3:p. 2-2]:

- Modal test outline to lead the user through the steps necessary to conduct a modal test.
- Easy editing of measurement location coordinates and geometry connectivity.
- Automatic communication with a spectrum analyzer via an IEEE-488 interface.
- Auto-configuration of spectrum analyzer.
- Transfer of measured FRFs from the spectrum analyzer to the program.
- Three different curvefitters: co/quad, circle, or rational fractional polynomial fit.
- Real-time animation of deflection shapes.
- Ability to plot undeformed/deformed geometry.
- Ability to plot measurements in real, imaginary, magnitude, phase, Bode or Nyquist format.

#### **F. EMODAL-PC MODAL TESTING FUNDAMENTALS**

##### **1. Test Procedures**

The EMODAL-PC program is a valuable tool for the analyst or designer wishing to analyze and solve noise and vibration problems of structures or mechanical systems. The most successful approaches to solving these problems usually utilize a combination of analysis and testing. In this vein, the EMODAL-PC program can be used by the analyst in the following general sequence of steps [Ref. 3:p. 6-2].

- Step 1: Identification of problem components or systems, usually done by operating vibration tests.
- Step 2: Perform experimental modal analysis, in which FRFs are measured and the system's modal parameters are identified.

- Step 3: Curvefitting and calibration of modal parameters to extract the mode shapes from the frequency response data.
- Step 4: Predictions of dynamic response, prior to structural modification analysis.

## 2. Modal Testing vs. Operating Deflection Shape Test

### a. Modal Testing

Modal testing can be considered the experimental, or measurement-taking approach, to determining a structure's or system's modal parameters. In conducting a modal test, the test structure is subjected to some externally-applied force. An FRF is obtained (actually estimated) from the simultaneous measurement of the exciting force and the response. The modal parameters are then extracted from a set of FRFs obtained at different locations on the structure.

### b. Operating Deflection Shape Test

An operating deflection shape test is the process of determining the vibration of some structure or system while under operating conditions. It is not a prediction of a system's dynamic behavior such as would be the result of a modal test, but instead it is a direct measure of the system's dynamic behavior. To conduct an operating deflection shape test, the structure (usually running machinery) vibrates due to self-induced forces. The forces generated by the structure are not measured as in modal testing. Instead, the vibration deflection shape is determined from a set of vibration

amplitude and phase measurements obtained at many different locations on the structure.

### **3. Curvefitting**

In general terms, curvefitting is the process of matching a set of data points obtained from FRF measurements to a mathematical expression based on a mass-spring-damper model. The simplest type of curvefit, the straight line curvefit, is recognized by most engineers as the result of a linear regression analysis. In the world of modal analysis, curvefitting is the first step in the parameter estimation process in which one (or a few) measurement is used to estimate the test structure's global modal parameters, specifically natural frequency, damping, modal mass and stiffness, which, at least in theory, are independent of position on the structure. This is then followed by the second step in the parameter estimation process, shape extraction, which uses all of the measurements obtained during the test to estimate the structure's mode shapes. Mode shape coefficients are known as local parameters because they do depend on the position on the structure.

#### **a. Coincident/Quadrature (Co/Quad) Fit**

There are numerous curvefitting techniques currently being used in modal analysis, each having its advantages and limitations. The EMODAL-PC program gives the user a choice of three of the more common curvefitting

techniques. The simplest modal analysis curvefit method available is the coincident/quadrature (co/quad) fit. This method, which is also called the peak pick method because it searches the measurement for the spectrum line with the maximum magnitude in the fit band, and uses this amplitude as an estimate of the shape coefficient, is used mainly for extracting operating deflection shapes, and for extracting mode shapes quickly to check the integrity of the data. Damping and other modal parameters are not estimated in this method. To use this method, all modes must be uncoupled and the resonance peaks must be well-separated.

#### ***b. Circle Fit***

The second type of curvefitting technique available in EMODAL-PC is the circle fit. This curvefit method is used primarily for extracting modal parameters when the modes are well-separated and when accurate estimates of damping are not required. It performs a least squared error algorithm which gives the shape coefficient, natural frequency and a rough estimate of the damping.

#### ***c. Rational Fraction Polynomial Fit***

The third type of curvefitting technique available in EMODAL-PC is the rational fraction polynomial fit. This curvefit method is a good, general purpose curvefitter that gives good results in most measurement cases. Its main uses include single DOF curvefitting on coupled modes (situation



when two or more modes are so close in frequency that it is difficult to distinguish between peaks) and multiple DOF curvefitting. The recommended procedure to use with this curvefit method is to attempt to fit modes using the single DOF fit, but if the results obtained are poor, move on to the multiple DOF fit. This is the recommended curvefitting technique to use if the most accurate results are desired.

#### **G. EMODAL-PC TEST PROCEDURE**

Conducting a modal analysis of a complex structure is not necessarily an easy task, but the use of modal analysis software decreases the level of difficulty. The EMODAL-PC software has a modal test outline window, obtained by pressing <?> from the EMODAL-PC main monitor, which leads the user through the entire process of setting up the test, taking measurements, extracting mode shapes, and storing all the data for future use. The test procedure, as presented in the modal test outline [Ref. 13:p. 2-4], is shown in Figure 13, and is defined in the sections that follow.

##### **1. Configuring the General Set-up**

The first step in conducting a modal test using the EMODAL-PC software is to configure the software for the test about to be conducted. It is in this first step that the user names the project and tells the program on what drive and directory to store the project and measurement files. The type of test to be conducted (either modal test or operating

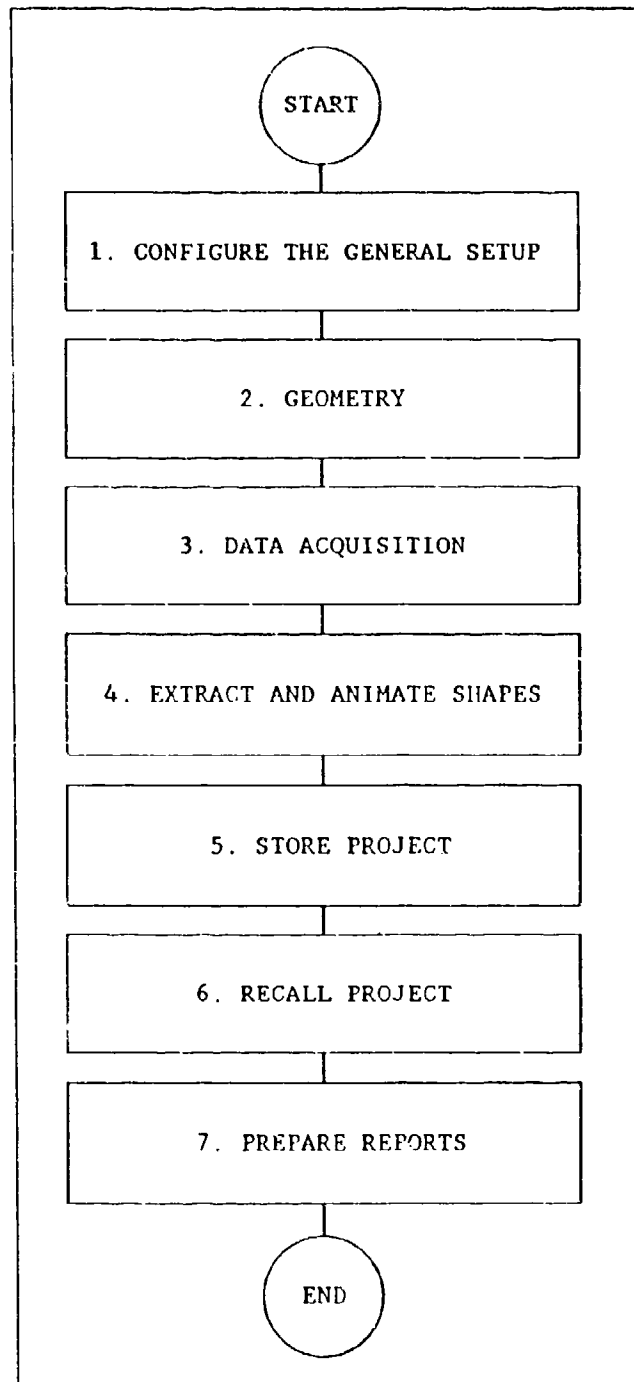


Figure 13 EMODAL-PC Test Procedure

deflection shape test) is chosen, as well as whether the output will be sent to the computer screen, a printer or a plotter. The most important input in this step is the analyzer GPIB address, without which the program cannot transfer measurements from the analyzer.

## **2. Geometry**

The second step of the modal test outline is the test structure geometry definition, in which the user enters the coordinates of the measurement locations on the test structure, as well as the links (or elements) connecting the measurement locations. There are three separate and necessary processes that take place in defining the structure's geometry within the modal analysis software. The first is the process of entering the x, y and z coordinates of all the measurement locations into the EMODAL-PC software. These measurement locations should be completely thought-out and defined by the analyst prior to entering them into the computer. The add function can be used to input additional measurement locations, should this be necessary, and the edit function can be used to modify or correct locations that have already been entered. Once all of the measurement locations are entered, the next step is to connect these locations together. The links, or elements, defined in this step are the instructions to the program to accurately draw the test structure. Like the coordinates commands, the add or insert function can be

used to enter new links and the edit function is used to modify or correct links that have already been entered. After all locations and links have been entered, the display geometry function should be selected to verify that the structure has been drawn correctly. This function allows the displayed structure to be translated or rotated about the axes, as well as magnified to verify small, hidden or complex parts of the structure. Once the displayed geometry has been verified, the analyst is ready to begin the actual conduct of the modal test.

### **3. Data Acquisition**

The data acquisition process is the most important part of a modal analysis, as only valid measurements will give valid test results. Obtaining valid measurements requires, as a minimum, that the analyst understand the operation of the spectrum analyzer in conjunction with the modal analysis software package, that the transducers being used have been calibrated correctly and can measure response in the frequency range of interest, and that the analyst use good, careful and consistent measurement technique. The correct procedure to use in acquiring data is to keep the reference transducer stationary throughout the test and move the response transducer(s) to various locations on the structure. The spectrum analyzer will then measure FRFs between the reference and response coordinates.

The first and most important step in the data acquisition process is transferring the measurements from the spectrum analyzer to the program. To accomplish this successfully, the data acquisition parameters of the EMODAL-PC software program must be configured properly. These parameters include the project name, reference coordinate, measurement drive and directory, calibration setup, and measurement auto-store feature. After the data acquisition setup has been completed, the analyzer can be properly configured. This can be a rather complex task as the SD-380 has 11 setup pages, each dedicated to a specific group of functions or modes, and discussed in detail in the SD-380 Signal Analyzer Operator's Manual [Ref. 14]. However, the data acquisition menu in the modal test outline contains a "Help setting up analyzer front panel" selection, which provides the user with some help in this setup procedure. Upon completion of analyzer setup, the "Read measurement" command can be chosen from the data acquisition menu, which results in the program prompting the user for the response coordinate for which a transfer function is desired. After the location and direction (for example, 1x) have been entered, the program will begin the data transfer process. When completed, the response transducer(s) may be moved to new locations, and the process continued until transfer functions have been obtained for all measurement locations of interest.

The second step in the data acquisition process is the storing and recalling of measurements. The recommended and easiest procedure for storing measurements is to have the program automatically store them when they are transferred from the spectrum analyzer. This is accomplished via the "Auto-store measurements=ON" parameter in the data acquisition setup menu. Otherwise, the measurements can be stored manually via the "Store measurement" command in the data acquisition menu. To recall a measurement previously stored, simply select the "Recall measurement" command from the data acquisition menu, and the program will prompt the user for a measurement number. The measurement number is the number the program assigns to a particular measurement, and is not necessarily the same as the location number on the test structure at which the measurement was made. To determine the measurement number assigned to a particular measurement, select the "Measurement directory" command from the data acquisition menu, enter the project name at the prompt, then enter "a" to obtain a directory listing of all measurements. When the measurement number of interest is determined, enter that number at the "Recall measurement" prompt, and the program will then recall that measurement.

The third step in the data acquisition process is viewing a measurement, usually performed after recalling a measurement. After selecting the "View measurement" command from the data acquisition menu, the program displays the

recalled measurement, along with commands which allow the user to magnify or expand the view, set up frequency bands for shape extraction and execute curvefits.

#### **4. Extract and Animate Shapes**

This is the step in a modal test where fit bands are set up and curvefits are performed, and mode shapes are extracted and animated. The purpose of setting up fit bands is to identify the frequencies of interest on a displayed measurement for which a curvefit will be conducted. The procedure for setting up fit bands and conducting a curvefit is simple and straight-forward, with the program prompting the user for the desired input. From the "View measurement and set up fit bands" option in the "Extract and Animate Shapes" menu, the <SETUP FIT> option should be selected. The program will prompt the user for a shape number (assigned in order of increasing frequency), curvefit type (either co/quad, circle or polynomial) and the upper and lower limits of the desired curvefit band. The user then selects the <CURVEFIT> option to execute the curvefit, for which the program displays a graph of the curvefit results in real, imaginary, log magnitude and Nyquist formats. After the curvefit, the user has the option of rejecting the curvefit or accepting it, if the results are satisfactory, and saving the results in the project's parameter table.

To extract shapes, each measurement on the disk is curvefit using the above process. For each fit, a shape coefficient is computed and saved in the project's shape file, from which the structure's mode shapes are generated. To initiate the shape extraction process, select the "Extract shapes" option from the "Extract and Animate Shapes" menu, then select the "Shape Extraction Setup" option and set the parameters as desired. The parameters include extract type (real or complex), reference coordinate, and measurement drive/directory. After these parameters have been set properly, select the "Extract shapes" option, and enter the measurement numbers for which extracted shape coefficients are desired. The program will recall these measurements from disk and curvefit them, resulting in the generation of the structure's mode shapes, which can then be animated.

The purpose of animating the shapes is to observe how the different measurement locations on the structure move, either in translation or rotation or both, as a result of the excitation force. After the shapes have been extracted, the user selects the "Animate shapes" option from the "Extract and Animate Shapes" menu, and enters the desired shape number. The program then displays the animation of the deformed structure. Numerous options are given in the animation menu, including the ability to zoom into different parts of the structure, rotate or translate the geometry, and display both the deformed and undeformed geometry on dual plots.



## 5. Store Project

All of the files associated with a particular project can be stored very easily by selecting the "Store Project" menu from the modal test outline. It is very important to store the project files immediately upon completion as the files will be lost if the program stops for any reason (e.g., power outage or user exiting the program without saving). There are actually six different types of project files that are stored when this option is selected. Each of these files, when saved, is given the same name as the project name chosen when the general setup is configured, but each has a different filename extension, as summarized below [Ref. 13:p. 2-61].

- Coordinates file - contains the x, y, and z coordinates of each measurement location; has extension .CRD
- Links file - contains the connectivity that draws the test structure; has extension .LNK
- Parameter table file - contains the fit band definitions and the modal parameters extracted by curvefitting; has extension .PAR
- Shapes file - contains the deflection shapes, arranged in columns by shape number; has extension .Sxx
- Setup menus file - contains program configuration parameter settings; has extension .SET
- Title file - contains the project title; has extension .TTL

Once all of the project files have been stored on the computer's hard disk, they can easily be copied on floppy disks for future reference.

## **6. Recall Project**

A previously stored project may need to be recalled back into computer memory for the purpose of reviewing the results or preparing reports. To recall a project, simply select the "Recall Project" step of the modal test outline and enter the project name at the prompt. The program will recall all of the project files and upon completion, will redisplay the modal test outline.

## **7. Prepare Reports**

This final step allows the user to print project files, produce plots of the undeformed and deformed structure, and produce plots of specific measurements, all in report-ready format. Prior to receiving any of these outputs from the program, however, the program must be correctly configured for the printer and/or plotter to be used. This is accomplished during the "General Program Setup" step. After verification of the correct configuration, and upon selection of the "Prepare Reports" option of the modal test outline, the user is given four options. The first option, "Print all files and tables", displays the print selection menu, from which the file or files to be printed are selected. The second option, "Plot shapes", will produce plots of either the undeformed or deformed structure, sending these plots either to the computer screen, or to an external printer or plotter. The third option, "Recall a measurement from disk", is used to

recall a specific measurement from the measurement directory for the purpose of plotting it. The fourth option, "Plot measurement", can then be selected to produce a plot of that measurement. However, the "Measurement plotting setup" menu must first be configured for plotting the measurement of interest. In this menu, the user selects plotting parameters including type of plot (real, imaginary, Bode, etc.), line type, title, x-axis and y-axis labels and divisions, scaling, and so on.

#### IV. CONCLUSIONS

##### A. ANALYTICAL MODELING

Within the scope of this investigation, the MSC/PAL2 finite element software is satisfactory for obtaining an analytical model of a missile system, and analytically determining its natural frequencies and mode shapes. The conduct of a normal modes analysis yields insight into the dynamic behavior of the structure under a wide variety of dynamic loading. Knowing the lowest few resonant frequencies and mode shapes provides the designer or analyst with information which may aid in identifying potential areas requiring redesign or possible interface problems between components of a system.

In the case of the AIM-9 missile, the material properties and mass distributions in the model obtained via the PAL2 software can be modified, based on the outcome of an experimental comprehensive modal test, to accurately represent the dynamic response of the actual missile. Once an accurate model is obtained, feasibility studies can easily be performed for the AIM-9 upgrades currently being planned. Specifically, the AIM-9M-8/9 program will retrofit existing missiles with an infrared counter-countermeasures feature, the AIM-9R will incorporate an imaging focal plane array to increase target

acquisition range while improving the missile's countermeasure and clutter rejection capabilities, and the year 2000 AIM-9X will feature an improved seeker, airframe, and warhead/fuze [Ref. 15]. All of these changes may affect the material's parameters and mass distributions, which can be readily modified in the finite element model.

The AIM-9 model developed in this investigation satisfactorily represents an actual missile; however, the material's parameters and mass distribution have yet to be matched to the natural frequencies and mode shapes determined by the dynamic modal measurement of an actual missile. Modal tests are planned for a follow-on phase of the investigation, at which point the model can easily be modified to reflect an accurate dynamic response by altering the mass locations, element stiffness or material properties.

## **B. EXPERIMENTAL MODELING**

While the GVT stand is satisfactory for longitudinal excitation of a test article, it is presently unsuitable for excitation of a test article in the lateral direction and must be structurally stiffened prior to the conduct of a comprehensive missile modal test.

Within the scope of this investigation, the modal analysis test setup and the associated methodologies established are satisfactory for conducting a comprehensive modal analysis of the AIM-9 missile system for the purpose of experimentally

determining the missile's transient response and modal parameters. The Fast Fourier Transform-based analyzer and the EMODAL-PC modal analysis software have been incorporated into the test setup, and are ready for transfer function measurement and subsequent determination of a structure's modal parameters.

### C. COMBINED TESTING AND ANALYSIS

The use of the analytical and experimental techniques in a complementary manner allows the designer or analyst to draw on the strengths of each technique to thoroughly and completely describe the dynamic response of a structure. By first determining a structure's dynamic response experimentally and then modifying the parameters in an analytical model to match that response, the analyst can use this analytical model to investigate different design options, sometimes called "what if?" studies, thereby avoiding the costly and time-consuming trial-and-error approach to solving vibration problems on the actual hardware. The "what if?" approach can be iterated on the model before any hardware modifications or upgrades are implemented.

## V. RECOMMENDATIONS

### A. ANALYTICAL MODELING

After completion of a modal test on the AIM-9 missile system, the analytical model should be modified to match the same mode shapes and natural frequencies observed experimentally. This can be accomplished by changing the missile's material properties, by re-distributing the lumped masses more evenly along the missile body, by adding stiffness to the front fins and by defining the missile's attachment points across an element vice at a single node point. The final and refined version of the analytical model, and hence the actual modifications made to the model, will depend entirely on the mode shapes and frequencies obtained via modal analysis.

Upon completion of an accurate AIM-9 missile analytical model, design iterations should be conducted for the planned AIM-9 upgrades to analytically determine the natural frequencies and mode shapes of these upgrades, in an effort to ascertain whether a flutter problem will exist between the P-3 wing (or the Orion II wing) and the upgraded AIM-9 missile. This can be accomplished primarily by modifying the model's mass distribution for the AIM-9M-8/9 and AIM-9R, and by

modifying the model's mass distribution, fin surfaces and possibly the shape of the body for the AIM-9X version.

## **B. EXPERIMENTAL MODELING**

The GVT stand needs to be stiffened laterally to allow excitation measurements in the lateral direction. The structure could be stiffened in several ways, including replacing the upper and lower 3/16" plates by thicker 1/2" plates, adding sandbags between the plates to provide damping, and adding transverse bracing between the three internal "I" beams.

The AIM-9 missile system should undergo a comprehensive modal test to completely determine its mode shapes and natural frequencies experimentally in both the longitudinal and lateral directions, using the missile modal analysis test setup completed in this investigation. The results of this comprehensive modal test should then be compared to the P-3 Orion and Orion II wing models to examine any potential flutter problems at any of the proposed wing carriage stations.

## **C. COMBINED TESTING AND ANALYSIS**

The facilities and methodologies discussed and used in this investigation of the AIM-9 missile's dynamic response should be utilized to conduct similar combined testing and analysis on other candidate missile systems, including the



AGM-65 Maverick and AGM-88 HARM, considered for incorporation into the P-3's weapon inventory.

## APPENDIX A

The MSC/PAL2 software code for the finite element model of the AIM-9 missile is given below. Explanatory comment statements were included where appropriate. For those PAL2 commands which could not be fully explained in a brief comment statement, the appropriate page of the MSC/PAL2 User's Manual [Ref. 6] is given, on which a detailed discussion of the command can be found.

TITLE - AIM9 GEOMETRY FILE

C

C DEFINE NODES FOR MISSILE BODY IN CYLINDRICAL COORDINATES

C

NODE 3

C

C NODE NUMBER, RADIUS, THETA, Z

C (Ref. 6:p. 3-92)

1 2.50, 0, 0 THROUGH 16 2.50, 337.5, 0

545 2.50, 0, 102 THROUGH 560 2.50, 337.5, 102

561 0, 0, 0

C

C GENERATE NODES OF MISSILE BODY AUTOMATICALLY

C (Ref. 6:p. 3-99)

NODE 22

1, 16, 560, 1, 16

NODE 22

1, 16, 560, 15, 16

C

C GENERATE NODES OF TAPERED NOSE CONE AUTOMATICALLY

C

NODE 3

562 2.50, 0, 103 THROUGH 577 2.50, 337.5, 103

594 1.25, 0, 112 THROUGH 609 1.25, 337.5, 112

NODE 22

562, 577, 609, 1, 16

NODE 22  
562, 577, 609, 15, 16

NODE 3  
610 0, 0, 113

C  
C CHANGE POSITION OF CERTAIN NODES TO ALLOW ATTACHMENT OF FINS  
C  
C MOVE SECOND & TENTH CONCENTRIC RINGS 1 INCH IN -Z DIRECTION  
C

NODE 1  
17 0, 0, -1 THROUGH 32 0, 0, -1

NODE 1  
145 0, 0, -1, THROUGH 160 0, 0, -1

C MOVE 34TH CONCENTRIC RING 2 INCHES IN THE -Z DIRECTION

NODE 1  
529 0, 0, -2, THROUGH 544 0, 0, -2

C  
C DEFINE NODE POINTS FOR FRONT FINS  
C

NODE 3

C  
C FIRST FRONT FIN  
C

611 2.75, 45, 93  
612 2.75, 45, 96  
613 2.75, 45, 97  
614 2.75, 45, 101.5  
615 7.00, 45, 97.25  
616 7.00, 45, 95  
617 7.00, 45, 94.5  
618 7.00, 45, 93  
619 11.25, 45, 93

C  
C SECOND FRONT FIN  
C

620 2.75, 135, 93  
621 2.75, 135, 96  
622 2.75, 135, 97  
623 2.75, 135, 101.5  
624 7.00, 135, 97.25  
625 7.00, 135, 95  
626 7.00, 135, 94.5  
627 7.00, 135, 93  
628 11.25, 135, 93

```

C
C THIRD FRONT FIN
C
629 2.75, 225, 93
630 2.75, 225, 96
631 2.75, 225, 97
632 2.75, 225, 101.5
633 7.00, 225, 97.25
634 7.00, 225, 95
635 7.00, 225, 94.5
636 7.00, 225, 93
637 11.25, 225, 93
C
C FOURTH FRONT FIN
C
638 2.75, 315, 93
639 2.75, 315, 96
640 2.75, 315, 97
641 2.75, 315, 101.5
642 7.00, 315, 97.25
643 7.00, 315, 95
644 7.00, 315, 94.5
645 7.00, 315, 93
646 11.25, 315, 93

C
C DEFINE NODE POINTS FOR REAR WINGS
C
NODE 3
C
C FIRST REAR WING
C
647 4.75, 45, 2
648 4.75, 45, 6
649 4.75, 45, 9
650 4.75, 45, 12
651 4.75, 45, 15
652 4.75, 45, 18
653 4.75, 45, 21
654 4.75, 45, 24
655 7.88, 45, 21
656 11.50, 45, 18
657 11.50, 45, 15
658 11.50, 45, 12
659 11.50, 45, 9
660 11.50, 45, 6
661 11.50, 45, 2
C
C SECOND REAR WING
C
662 4.75, 135, 2

```

663 4.75, 135, 6  
664 4.75, 135, 9  
665 4.75, 135, 12  
666 4.75, 135, 15  
667 4.75, 135, 18  
668 4.75, 135, 21  
669 4.75, 135, 24  
670 7.88, 135, 21  
671 11.50, 135, 18  
672 11.50, 135, 15  
673 11.50, 135, 12  
674 11.50, 135, 9  
675 11.50, 135, 6  
676 11.50, 135, 2

C

C THIRD REAR WING

C

677 4.75, 225, 2  
678 4.75, 225, 6  
679 4.75, 225, 9  
680 4.75, 225, 12  
681 4.75, 225, 15  
682 4.75, 225, 18  
683 4.75, 225, 21  
684 4.75, 225, 24  
685 7.88, 225, 21  
686 11.50, 225, 18  
687 11.50, 225, 15  
688 11.50, 225, 12  
689 11.50, 225, 9  
690 11.50, 225, 6  
691 11.50, 225, 2

C

C FOURTH REAR WING

C

692 4.75, 315, 2  
693 4.75, 315, 6  
694 4.75, 315, 9  
695 4.75, 315, 12  
696 4.75, 315, 15  
697 4.75, 315, 18  
698 4.75, 315, 21  
699 4.75, 315, 24  
700 7.88, 315, 21  
701 11.50, 315, 18  
702 11.50, 315, 15  
703 11.50, 315, 12  
704 11.50, 315, 9  
705 11.50, 315, 6  
706 11.50, 315, 2

```

C
C ENTER MATERIAL PROPERTIES
C
C YOUNG'S MODULUS, SHEAR MODULUS, MASS DENSITY, POISSON'S
C RATIO, TENSILE YIELD STRESS
C (Ref. 6:p. 3-83)
MAT 30.0E6, 12.0E6, 7.76E-4, 0.25, 30.0E3
C
C DEFINE PLATE TYPE FOR MISSILE BODY
C
C QUADRILATERAL PLATE ELEMENT, N1, N2, THICKNESS
C (Ref. 6:p. 3-113)
QUAD 1, 1, 0.25
C
C GENERATE QUAD PLATE ELEMENTS ON MISSILE BODY AUTOMATICALLY
C (Ref. 6:p. 3-56)
GENERATE CONNECTS 1, 16, 560, 1, 16
GENERATE CONNECTS 1, 16, 560, 15, 16
GENERATE CONNECTS 562, 577, 609, 1, 16
GENERATE CONNECTS 562, 577, 609, 15, 16
C
C DEFINE REAR MISSILE PLATE AND CONNECTIVITY
C
C TRIANGULAR PLATE ELEMENT, N1, N2, THICKNESS
C (Ref. 6:p. 3-121)
TRI 1, 2, 0.25
C
C CONNECT NODE A TO NODE B TO NODE C
C (Ref. 6:p. 3-27)
CON 1 TO 2 TO 561
CON 2 TO 3 TO 561
CON 3 TO 4 TO 561
CON 4 TO 5 TO 561
CON 5 TO 6 TO 561
CON 6 TO 7 TO 561
CON 7 TO 8 TO 561
CON 8 TO 9 TO 561
CON 9 TO 10 TO 561
CON 10 TO 11 TO 561
CON 11 TO 12 TO 561
CON 12 TO 13 TO 561
CON 13 TO 14 TO 561
CON 14 TO 15 TO 561
CON 15 TO 16 TO 561
CON 16 TO 1 TO 561
C
C DEFINE PLATE CAP CONNECTIVITY OF NOSECONE
C
CON 594 TO 595 TO 610
CON 595 TO 596 TO 610
CON 596 TO 597 TO 610

```

CON 597 TO 598 TO 610  
 CON 598 TO 599 TO 610  
 CON 599 TO 600 TO 610  
 CON 600 TO 601 TO 610  
 CON 601 TO 602 TO 610  
 CON 602 TO 603 TO 610  
 CON 603 TO 604 TO 610  
 CON 604 TO 605 TO 610  
 CON 605 TO 606 TO 610  
 CON 606 TO 607 TO 610  
 CON 607 TO 608 TO 610  
 CON 608 TO 609 TO 610  
 CON 609 TO 594 TO 610  
 C  
 C CONNECT BODY TO NOSECONE  
 C  
 QUAD 1, 1, 0.25  
 C  
 C CONNECT NODE A TO NODE B TO NODE C TO NODE D  
 C (CONNECT COUNTERCLOCKWISE)  
 C (Ref. 6:p. 3-27)  
 CON 545 TO 562 TO 563 TO 546  
 CON 546 TO 563 TO 564 TO 547  
 CON 547 TO 564 TO 565 TO 548  
 CON 548 TO 565 TO 566 TO 549  
 CON 549 TO 566 TO 567 TO 550  
 CON 550 TO 567 TO 568 TO 551  
 CON 551 TO 568 TO 569 TO 552  
 CON 552 TO 569 TO 570 TO 553  
 CON 553 TO 570 TO 571 TO 554  
 CON 554 TO 571 TO 572 TO 555  
 CON 555 TO 572 TO 573 TO 556  
 CON 556 TO 573 TO 574 TO 557  
 CON 557 TO 574 TO 575 TO 558  
 CON 558 TO 575 TO 576 TO 559  
 CON 559 TO 576 TO 577 TO 560  
 CON 560 TO 577 TO 562 TO 545  
 C  
 C DEFINE FRONT FIN PLATE TYPE AND CONNECTIVITY  
 C  
 C FIRST FIN  
 C  
 C QUAD PLATE TO ATTACH FIN TO MISSILE BODY  
 C  
 QUAD 1, 1, 0.35  
 CON 515 TO 531 TO 613 TO 612  
 C  
 C FIN CONSTRUCTION  
 C  
 QUAD 1, 1, 0.10  
 CON 611 TO 612 TO 617 TO 618

CON 612 TO 613 TO 616 TO 617  
 CON 613 TO 614 TO 615 TO 616  
 TRI 1, 2, 0.10  
 CON 619 TO 618 TO 617  
 CON 619 TO 617 TO 616  
 CON 619 TO 616 TO 615  
 C  
 C SECOND FRONT FIN  
 C  
 QUAD 1, 1, 0.35  
 CON 519 TO 535 TO 622 TO 621  
 QUAD 1, 1, 0.1  
 CON 620 TO 621 TO 626 TO 627  
 CON 621 TO 622 TO 625 TO 626  
 CON 622 TO 623 TO 624 TO 625  
 TRI 1, 2, 0.1  
 CON 628 TO 627 TO 626  
 CON 628 TO 626 TO 625  
 CON 628 TO 625 TO 624  
 C  
 C THIRD FRONT FIN  
 C  
 QUAD 1, 1, 0.35  
 CON 523 TO 539 TO 631 TO 630  
 QUAD 1, 1, 0.1  
 CON 629 TO 630 TO 635 TO 636  
 CON 630 TO 631 TO 634 TO 635  
 CON 631 TO 632 TO 633 TO 634  
 TRI 1, 2, 0.1  
 CON 637 TO 636 TO 635  
 CON 637 TO 635 TO 634  
 CON 637 TO 634 TO 633  
 C  
 C FOURTH FRONT FIN  
 C  
 QUAD 1, 1, 0.35  
 CON 527 TO 543 TO 640 TO 639  
 QUAD 1, 1, 0.1  
 CON 638 TO 639 TO 644 TO 645  
 CON 639 TO 640 TO 643 TO 644  
 CON 640 TO 641 TO 642 TO 643  
 TRI 1, 2, 0.1  
 CON 646 TO 645 TO 644  
 CON 646 TO 644 TO 643  
 CON 646 TO 643 TO 642  
 C  
 C DEFINE MATERIAL PROPERTIES FOR REAR WINGS  
 C  
 MAT 10E6, 4E6, 2.59E-4, 0.25, 10E3



C  
 C DEFINE REAR WING PLATE TYPE AND CONNECTIVITY  
 C  
 C FIRST REAR WING  
 C  
 QUAD 1, 1, 0.5  
 CON 19 TO 35 TO 648 TO 647  
 CON 647 TO 648 TO 660 TO 661  
 CON 35 TO 51 TO 649 TO 648  
 CON 648 TO 649 TO 659 TO 660  
 CON 51 TO 67 TO 650 TO 649  
 CON 649 TO 650 TO 658 TO 659  
 CON 67 TO 83 TO 651 TO 650  
 CON 650 TO 651 TO 657 TO 658  
 CON 83 TO 99 TO 652 TO 651  
 CON 651 TO 652 TO 656 TO 657  
 CON 99 TO 115 TO 653 TO 652  
 CON 652 TO 653 TO 655 TO 656  
 CON 115 TO 131 TO 654 TO 653  
 TRI 1, 2, 0.5  
 CON 654 TO 131 TO 147  
 CON 655 TO 653 TO 654  
 C  
 C SECOND REAR WING  
 C  
 QUAD 1, 1, 0.5  
 CON 23 TO 39 TO 663 TO 662  
 CON 662 TO 663 TO 675 TO 676  
 CON 39 TO 55 TO 664 TO 663  
 CON 663 TO 664 TO 674 TO 675  
 CON 55 TO 71 TO 665 TO 664  
 CON 664 TO 665 TO 673 TO 674  
 CON 71 TO 87 TO 666 TO 665  
 CON 665 TO 666 TO 672 TO 673  
 CON 87 TO 103 TO 667 TO 666  
 CON 666 TO 667 TO 671 TO 672  
 CON 103 TO 119 TO 668 TO 667  
 CON 667 TO 668 TO 670 TO 671  
 CON 119 TO 135 TO 669 TO 668  
 TRI 1, 2, 0.5  
 CON 669 TO 135 TO 151  
 CON 670 TO 668 TO 669  
 C  
 C THIRD REAR WING  
 C  
 QUAD 1, 1, 0.5  
 CON 27 TO 43 TO 678 TO 677  
 CON 677 TO 678 TO 690 TO 691  
 CON 43 TO 59 TO 679 TO 678  
 CON 678 TO 679 TO 689 TO 690  
 CON 59 TO 75 TO 680 TO 679

CON 679 TO 680 TO 688 TO 689  
 CON 75 TO 91 TO 687 TO 680  
 CON 680 TO 681 TO 687 TO 688  
 CON 91 TO 107 TO 682 TO 681  
 CON 681 TO 682 TO 686 TO 687  
 CON 107 TO 123 TO 683 TO 682  
 CON 682 TO 683 TO 685 TO 686  
 CON 123 TO 139 TO 684 TO 683  
 TRI 1, 2, 0.5  
 CON 684 TO 139 TO 155  
 CON 685 TO 683 TO 684  
 C  
 C FOURTH REAR WING  
 C  
 QUAD 1, 1, 0.5  
 CON 31 TO 47 TO 693 TO 692  
 CON 692 TO 693 TO 705 TO 706  
 CON 47 TO 63 TO 694 TO 693  
 CON 693 TO 694 TO 704 TO 705  
 CON 63 TO 79 TO 695 TO 694  
 CON 694 TO 695 TO 703 TO 704  
 CON 79 TO 95 TO 696 TO 695  
 CON 695 TO 696 TO 702 TO 703  
 CON 95 TO 111 TO 697 TO 696  
 CON 696 TO 697 TO 701 TO 702  
 CON 111 TO 127 TO 698 TO 697  
 CON 697 TO 698 TO 700 TO 701  
 CON 127 TO 143 TO 699 TO 698  
 TRI 1, 2, 0.5  
 CON 699 TO 143 TO 159  
 CON 700 TO 698 TO 699  
 C  
 C DEFINE ATTACHMENT POINTS  
 C  
 C AT ATTACHMENT POINTS, ALLOW ROTATION ABOUT X AXIS ONLY  
 C (Ref. 6:p. 3-124)  
 ZERO 1  
 TA 85, 213, 373  
 RY 85, 213, 373  
 RZ 85, 213, 373  
 C  
 C DEFINE POINT MASSES FOR SEPARATE MISSILE COMPONENTS  
 C  
 C MASS COMMAND, NODE NUMBER, MASS OF COMPONENT  
 C  
 C MASS OF MOTOR COMPONENT (TWO POINT MASSES SINCE 70" LONG)  
 C  
 MASS 141, 0.128  
 MASS 269, 0.128

C  
C MASS OF WARHEAD COMPONENT  
C  
MASS 413, 0.0538  
C  
C MASS OF TARGET DETECTOR COMPONENT  
C  
MASS 477, 0.0233  
C  
C MASS OF GUIDANCE AND CONTROL COMPONENT  
C  
MASS 557, 0.0673  
END

## APPENDIX B

Figures 14 through 19 depict the first six mode shapes for the AIM-9 missile model constructed in this investigation. They represent the missile's deformations at its first six natural frequencies of 29.65, 50.79, 52.97, 57.36, 57.89, and 58.80 hz. It should be noted that these frequencies, and hence these mode shapes, are not necessarily representative of the actual missile response. These results were obtained mathematically, strictly from the model geometry, mass density, and additional lumped masses representing the individual missile components. The figures, presented in the optimum planar representation for viewing the responses, show still frames of the animated mode shapes: the first view showing the upper limit of deformation, the second showing the undeformed structure, and the third showing the lower limit of deformation. While not totally obvious from the figures, the missile model exhibited the following modal responses.

- Mode 1 (29.65 hz) - whole body z-axis rotation about attachment points with some bending in x-z plane
- Mode 2 (50.79 hz) - bending mode in y-z plane with slight front fin z-axis rotation
- Mode 3 (52.97 hz) - bending mode in x-z plane with slight front fin z-axis rotation
- Mode 4 (57.36 hz) - front fin z-axis rotation only; no body motion
- Mode 5 (57.89 hz) - front fin z-axis rotation with some body torsion in the x-y plane

- Mode 6 (58.80 hz) - front fin z-axis rotation with some body bending in the y-z plane and torsion in the x-y plane

The figures, showing what appear to be strange contortions of the front fins, are actually gross exaggerations arising from two possible phenomena. The first is the fact that the front fins are attached to the missile body only by a thin quadrilateral plate element, resulting in a lack of stiffness for these fins. The second is the fact that scaling for each mode shape in a normal modes analysis is arbitrary since no external load is applied; only the relative shape is important. The deformations of the front fins in these modes was actually less than 0.3 inches.

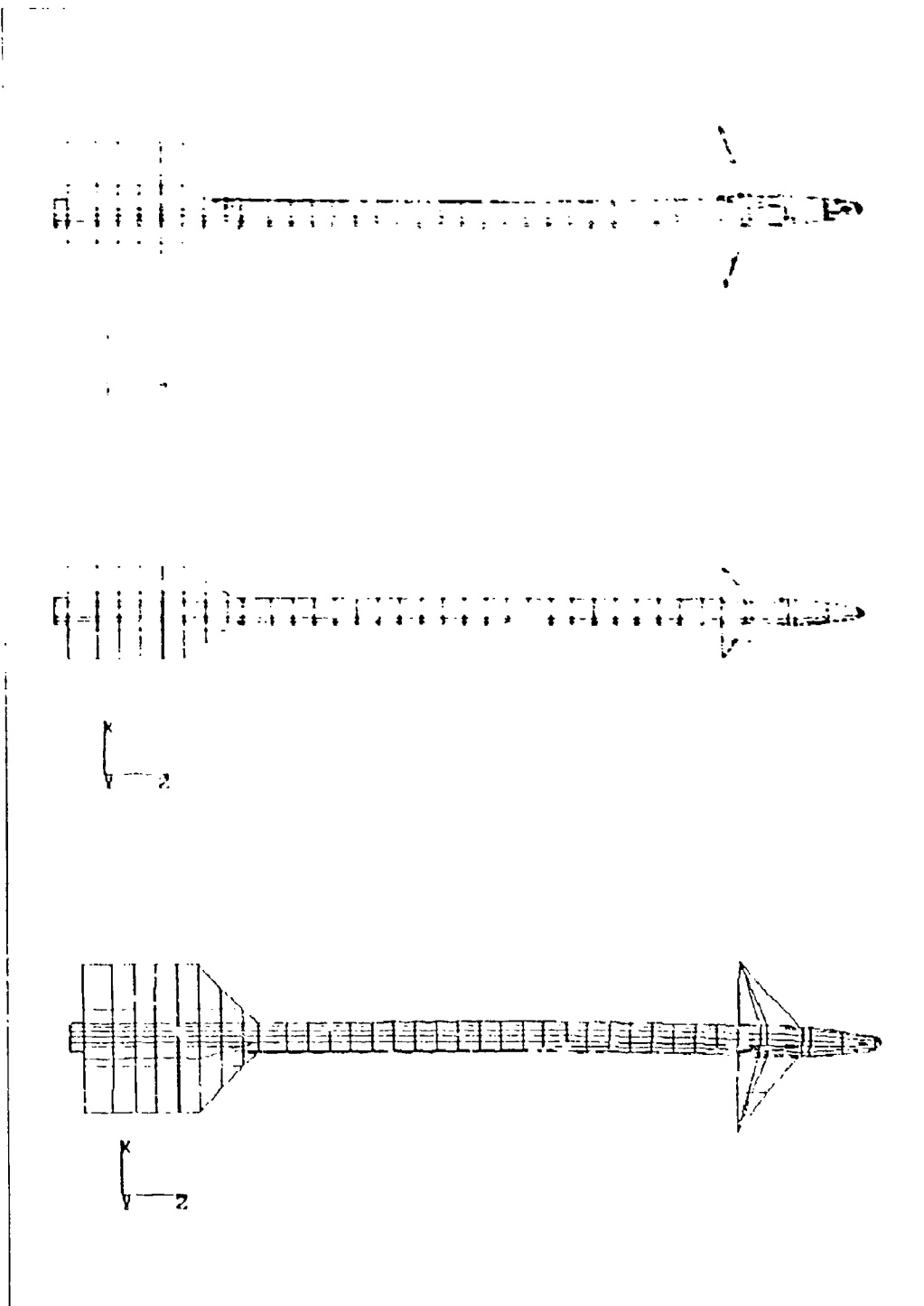


Figure 14 AIM-9 Missile Model Mode 1 Response

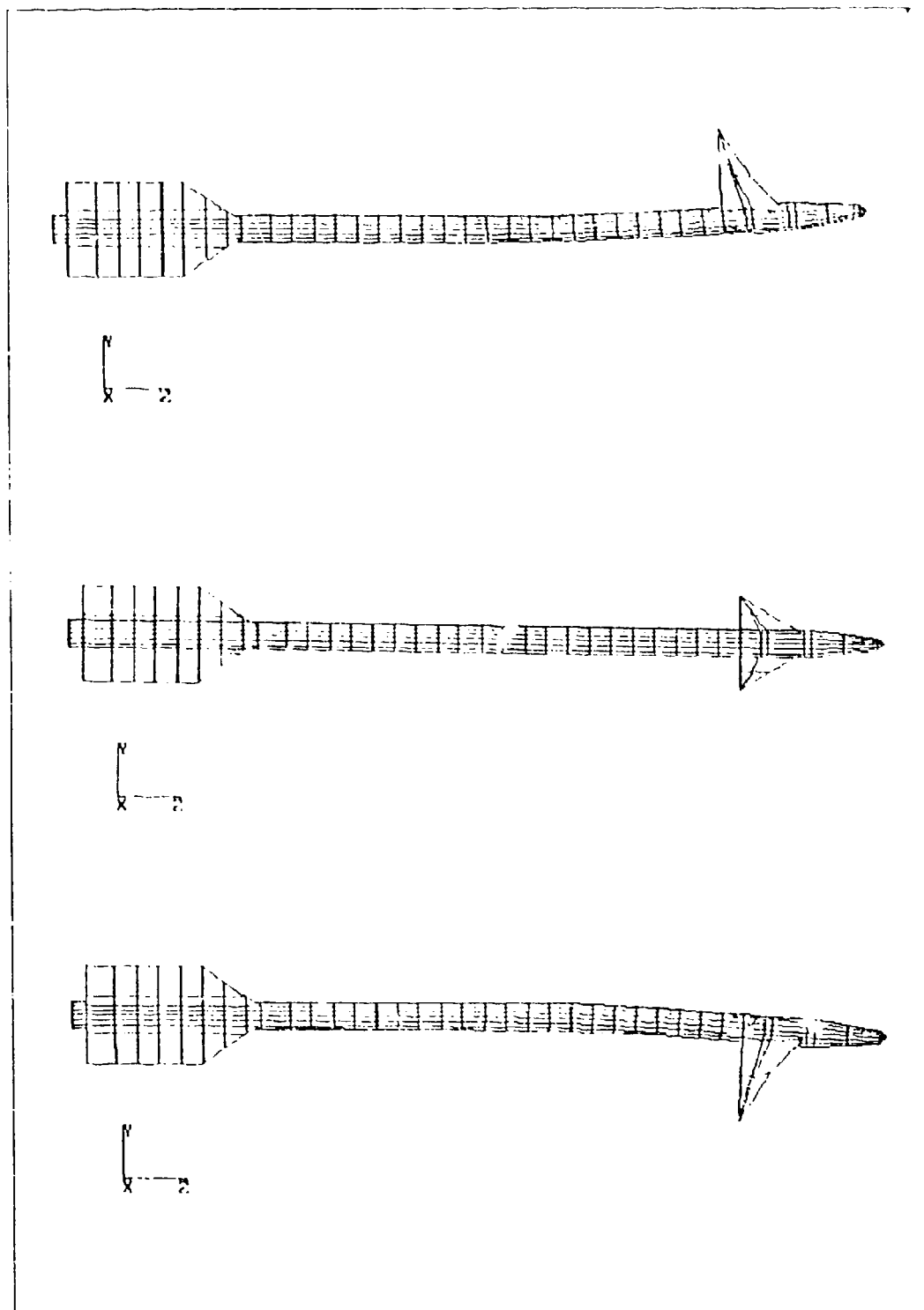


Figure 15 AIM-9 Missile Model Mode 2 Response

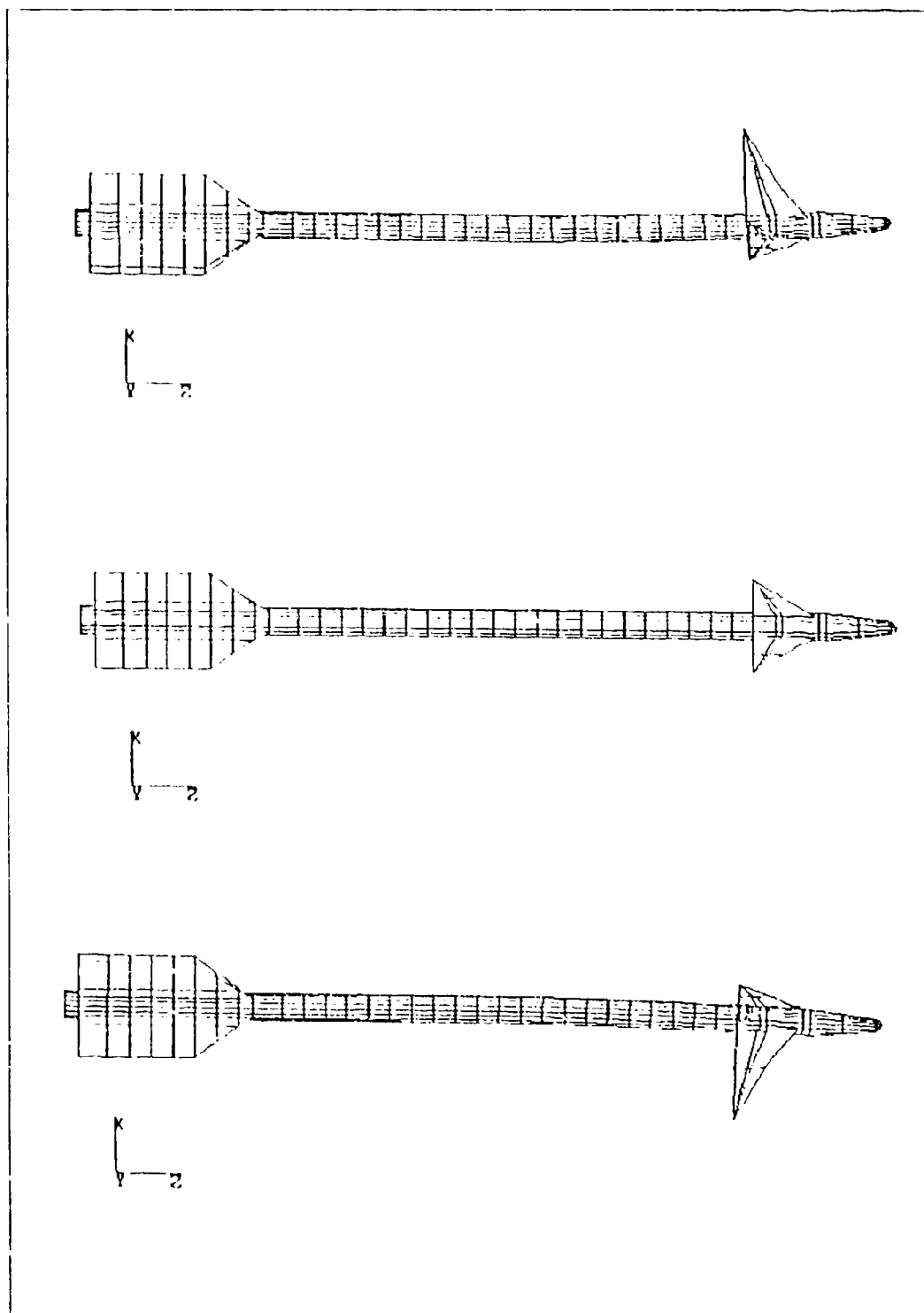


Figure 16 AIM-9 Missile Model Mode 3 Response



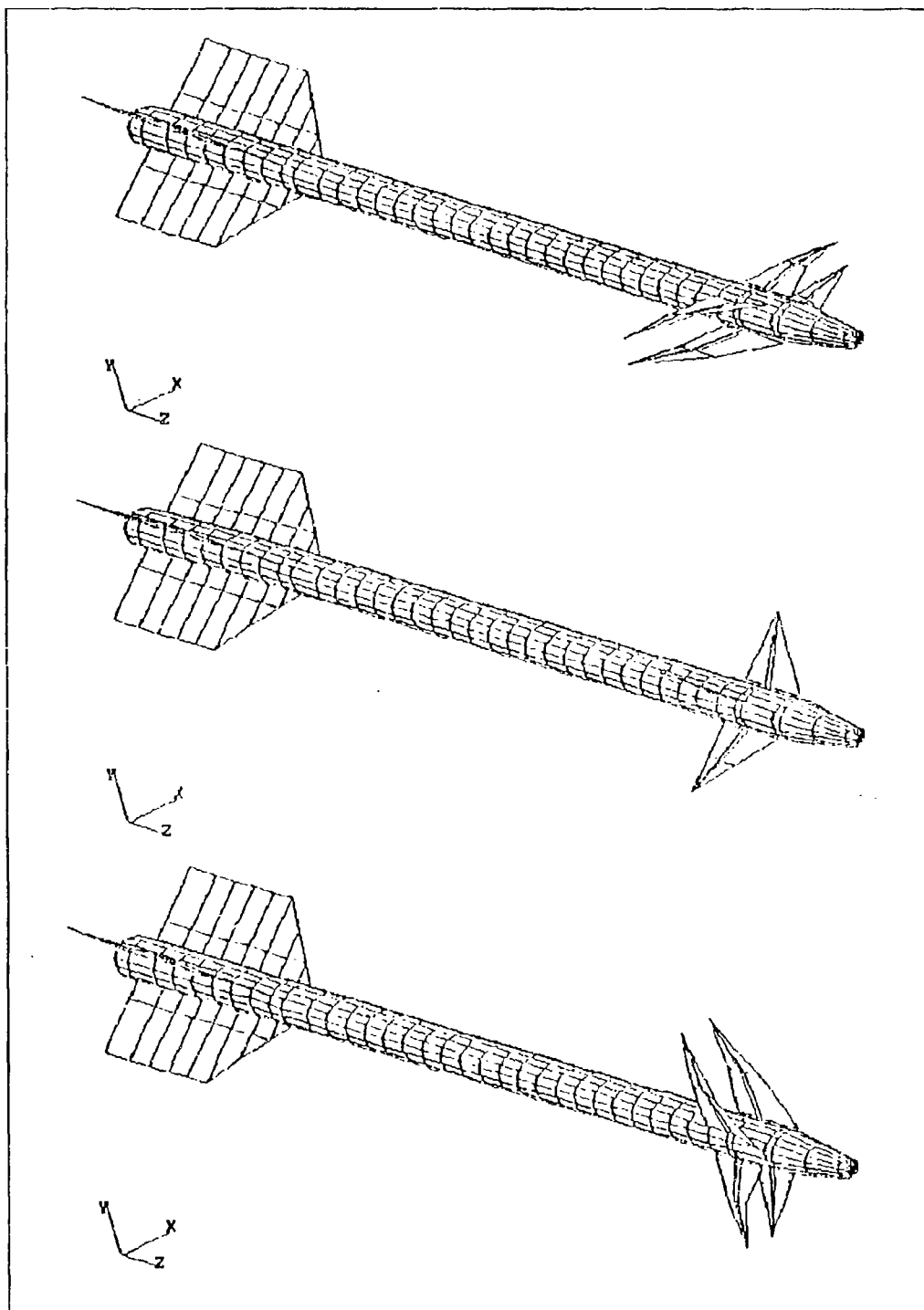


Figure 17 AIM-9 Missile Model Mode 4 Response

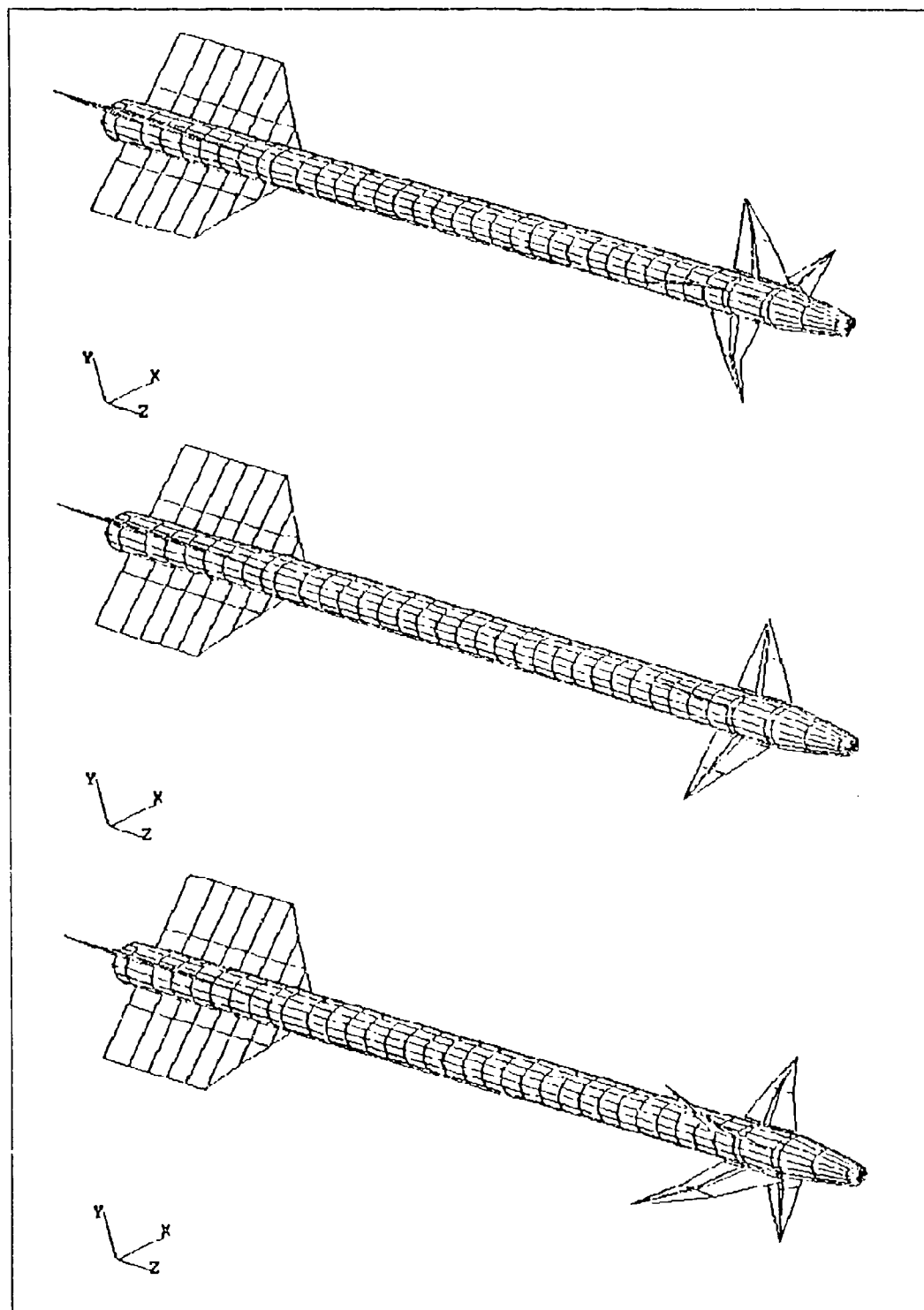


Figure 18 AIM-9 Missile Model Mode 5 Response

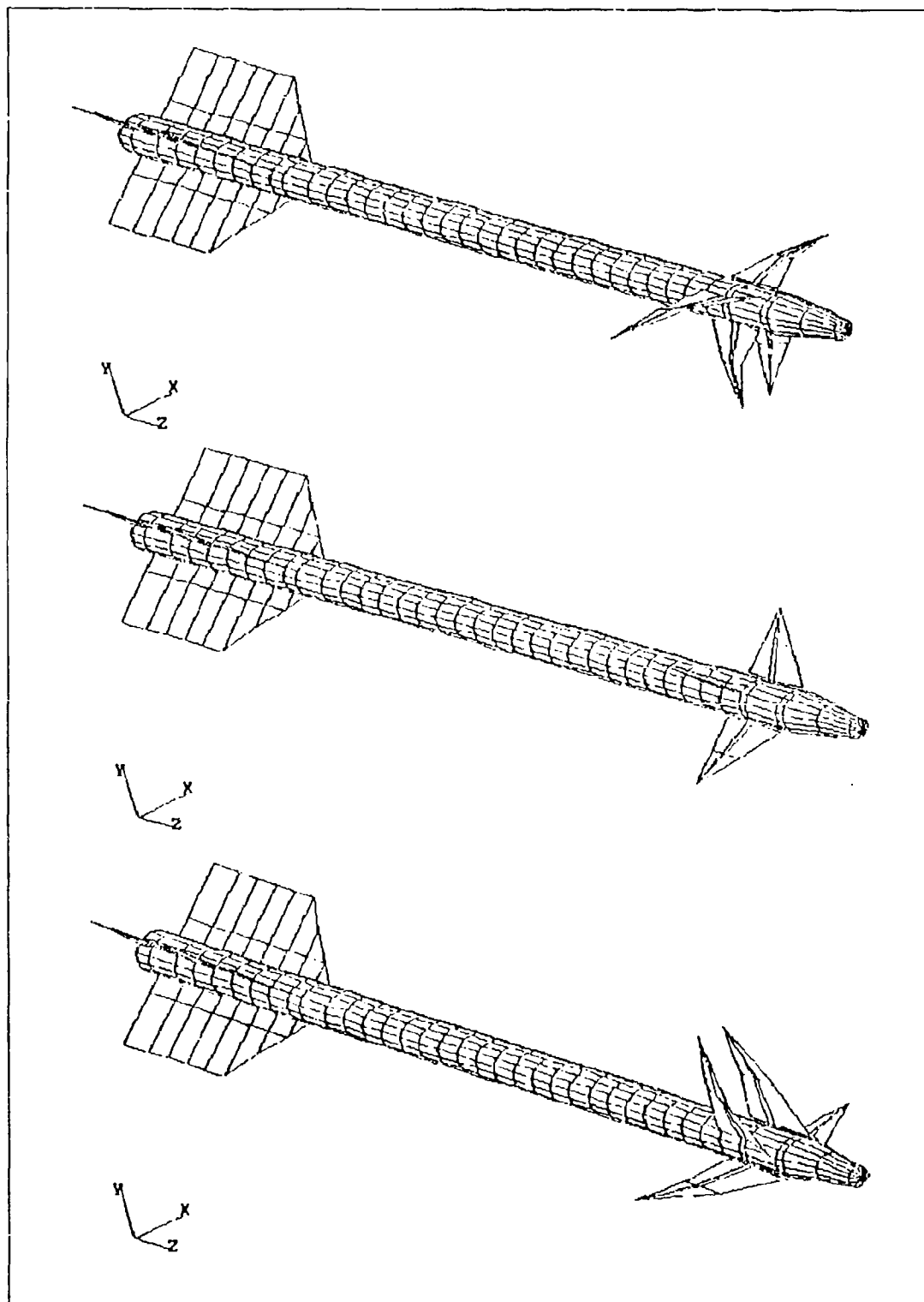


Figure 19 AIM-9 Missile Model Mode 6 Response

## APPENDIX C

### TRADE-OFF STUDIES

Several studies were conducted on the basic AIM-9 model in an attempt to determine the effects of changing the location of the point masses and the material properties of the structure. The results of these trade-off studies are given in the five cases discussed below in the order in which they were conducted, with the current AIM-9 model being Case V. For each of the cases, the first six mode shapes and their corresponding natural frequencies are described, except for Case II, in which only the first four mode shapes were determined. In studying these cases, one can see that changing the mass location or material properties can change both the mode shapes and the resonant frequencies, sometimes quite drastically.

**Case I** - missile body only with point masses located on top of missile body.

- Mode 1 (50.02 Hz) - bending in x-z plane
- Mode 2 (53.36 Hz) - bending in y-z plane
- Mode 3 (71.39 Hz) - rotation about z-axis
- Mode 4 (209.60 Hz) - complex bending in y-z plane
- Mode 5 (262.70 Hz) - complex bending in y-z plane and body torsion in x-y plane

- Mode 6 (291.80 hz) - complex bending in y-z plane and body torsion in x-y plane

**Case II** - missile body only with masses located on bottom of missile body but in same z-axis location.

- Mode 1 (33.42 hz) - rotation about z-axis, with bending in x-z plane
- Mode 2 (53.02 hz) - bending in y-z plane
- Mode 3 (59.75 hz) - bending in x-z plane
- Mode 4 (140.60 hz) - complex bending in y-z plane

**Case III** - full AIM-9 model with fins and wings, and five point masses located on bottom of missile body.

- Mode 1 (24.56 hz) - rotation about z-axis (about the attachment points)
- Mode 2 (50.80 hz) - bending in y-z plane
- Mode 3 (50.85 hz) - bending in x-z plane with some body torsion in x-y plane
- Mode 4 (57.36 hz) - front fin rotation about z-axis; no body motion
- Mode 5 (57.72 hz) - front fin rotation about z-axis with some body torsion in x-y plane
- Mode 6 (58.70 hz) - front fin rotation about z-axis with some body torsion in x-y plane and bending in x-z plane

**Case IV** - full AIM-9 model with ten point masses located on top and bottom of missile body at the same z-axis location.

- Mode 1 (26.68 hz) - rotation about z-axis (about the attachment points)
- Mode 2 (50.58 hz) - bending in x-z plane
- Mode 3 (51.01 hz) - bending in y-z plane

- Mode 4 (57.36 hz) - front fin rotation about z-axis; no body motion
- Mode 5 (57.57 hz) - front fin rotation about z-axis with some body torsion in x-y plane
- Mode 6 (58.71 hz) - front fin rotation about z-axis with some body torsion in x-y plane and bending in x-z plane

**Case V** - full AIM-9 model with five point masses located on bottom of missile body and the rear wings changed from steel to aluminum.

- Modes given previously in Appendix B.

## **APPENDIX D**

### **MODAL TEST SYSTEM TEST EQUIPMENT**

#### **Scientific-Atlanta Spectrum Analyzer Model SD-380**

The Model SD-380 is a microprocessor-based one, two or four channel signal analyzer. It measures the signal(s) present at one, two or four BNC connectors located on the rear panel, and in the case of the two and four channel options, the relationship between two signals. In addition to standard signal analysis capabilities, it is equipped with a built-in waterfall (or cascade) feature, a 56K word input memory, an IEEE interface, and the ability to perform single/double differentiation/integration. Additional specifications and information on the SD-380 are supplied by Scientific-Atlanta [Ref. 14].

#### **COMPAQ Deskpro 386/25e Model 60 Computer with EMODAL Software**

The Deskpro 386/25e includes a 25 Mhz 386 microprocessor, four (Mb) of random access memory, a 60 Mb hard disk drive, a high-density 1.2 Mb, 5 1/4 inch floppy disk drive, and a high-density 1.44 Mb, 3 1/2 inch floppy disk drive. It comes equipped with an enhanced keyboard, a COMPAQ Video Graphics monitor, and a mouse.

The software used in this test setup is Entek Scientific Corporation's EMODAL modal analysis software package. It can be used as a stand-alone modal testing tool, or as a tool to collect and process data. It has been designed to run with or without peripheral devices; however, a spectrum analyzer must be connected when acquiring measurement data. To allow the EMODAL software to communicate with the spectrum analyzer, the Deskpro 386/25e has had a National Instruments AT-GPIB board installed, which supplies the necessary IEEE-488 interface. More specific information on the EMODAL software can be obtained from the EMODAL-PC Instruction Manual [Ref. 3].

#### **Bruel & Kjaer (B&K) Exciter Type 4801**

The Type 4801 exciter was designed to be combined with any one of several different exciter heads, resulting in a complete exciter (or shaker) assembly, capable of generating up to a 100 pound excitation force. It is not considered a portable shaker with its rather substantial weight of 180 pounds, but it can be positioned at any angle to provide optimum flexibility and use. With the general purpose head, the Type 4801 exciter is capable of providing displacements up to 0.5 inches. It has an internal natural frequency of 7200 hz, with the base having a natural frequency of 10-14 hz. More detailed information can be found in the manufacturer's Instructions and Applications booklet [Ref. 16] and Vibration Exciter System V Instruction Manual [Ref. 17].



### **B&K General Purpose Exciter Head Type 4812**

The Type 4812 general purpose head was designed to provide a low distortion operating band from DC to 10,000 hz when used with the Type 4801 exciter and the Type 2707 amplifier. The 'g' loads available for testing vary with the test frequency and run from a low of zero in the DC range to 100g at 150-160 hz. This general purpose impedance head has both a force transducer and a response transducer incorporated into it, and is used to obtain driving point measurements. More detailed information can be found in the manufacturer's Instructions and Applications booklet [Ref. 18].

### **B&K Power Amplifier Type 2707**

The Type 2707 power amplifier was designed to provide proper power and protection to the Type 4801 exciter. Protective circuits incorporated into the power amplifier include power phase protection, signal ground fault, overtemperature, displacement, overcurrent and waveform clipping indications. More detailed information can be found in the manufacturer's Instruction Manual for Power Amplifier Type 2707 [Ref. 19].

### **PCB Piezoelectronics Flexcel Modal Accelerometer Model 336A**

The Model 336A modal accelerometer functions to transfer vibratory motion into a measurable voltage signal for convenient display and analysis. It is a low-mass

accelerometer which serves to minimize mass-loading effects. Characteristics of this accelerometer important in modal analysis applications include high output, low noise, matched phase response, low transient thermal sensitivity, and most importantly for the missile modal test, good low frequency response, provided by a composite construction which isolates from ground and thermal transients. Since this accelerometer contains a built-in microelectronic charge-to-voltage amplifier, a constant current power unit is required to operate it. Specifics concerning the Model 336A include a frequency range of 1-2000 hz, a sensitivity of 1000 mV/g, and a resonant frequency (mounted) of greater than 9 khz. More detailed information can be found in the manufacturer's operating instructions [Ref. 20].

#### **PCB Piezoelectronics Dual-Mode Charge Amplifier Model 464A**

The Model 464A is a dual-mode device which can be operated either as a charge amplifier (for transducers without built-in microelectronics) or as an Integrated-Circuit-Piezoelectric (ICP) constant current power source (for transducers with built-in microelectronics). Operation is simple in that switching to the mode of operation (charge or ICP), dialing in the transducer sensitivity, and switching to the desired range is all that is required. The Model 464A has a frequency response from near DC to 100 khz. For this test setup, a plug-in 180 khz low-pass filter was installed, as this was all

that was available at the time of the test setup. However, the filters are available in varying sizes, ranging from 1 khz to 180 khz. More detailed information can be found in the manufacturer's operating instructions [Ref. 21].

#### **PCB Piezoelectronics Mechanical Impedance Sensor Model 288A11**

The Model 288A11 is a low mass, combination force transducer and accelerometer which functions to transfer both the excitation (force) and the response (motion) into electrical signals. The built-in microelectronic amplifier converts the high impedance signals generated by the quartz element force gauge and accelerometer to relatively low impedance voltage signals. Since this sensor contains a built-in amplifier, a constant current power unit is required to operate it. Characteristics of this sensor include low mass (only 45 grams), nominal sensitivity of 100 mV/g (motion) and 1000 mV/g (force), and frequency range of 5-5000 hz. More detailed information can be found in the manufacturer's operating instructions [Ref. 22].

## LIST OF REFERENCES

1. Hollyer, J. B., *Ground Vibration Characterization of a Missile System for Flutter Energy Definition*, Master's Thesis, Naval Postgraduate School, Monterey, California, June 1990.
2. *Structural Measurement Systems, Basic Modal Testing and Analysis Techniques*, 1991.
3. Entek Scientific Corporation, *EMODAL-PC Instruction Manual*, 1988.
4. Bramer, T.F.C., and others, *Basic Vibration Control*, Sound Research Laboratories Limited, pp. 117-121, 1977.
5. MacNeal-Schwendler Corp., *MSC/PAL2 Advanced Stress and Vibration Analysis Version 4.0 Reference Manual*, 1990.
6. MacNeal-Schwendler Corp., *MSC/PAL2 Advanced Stress and Vibration Analysis Version 4.0 User's Manual*, 1990.
7. Private communication between Ken Ranger, MSC and the author, 7 August 1991.
8. Richardson, Mark, *Modal Analysis Using Digital Test Systems*, Hewlett-Packard Company.
9. Spectral Institute, *Modal Analysis Training Course and Application Notes*, p. 2-6, 1991.
10. American Society of Mechanical Engineers, *Vibration Testing - Instrumentation and Data Analysis*, pp. 37-38, 1975.
11. PCB Piezoelectronics, Inc., *Petro Wax Model 080A24 Instructions*.
12. Corelli, D. and Brown, D.L., "Impact Testing Considerations", Excerpt from ENTEK Scientific Corporation Modal Analysis Training Course Notes, undated.
13. Entek Scientific Corporation, *EMODAL-PC Learning Guide*, 1988.
14. Scientific Atlanta, *SD-380 Signal Analyzer Operator's Manual*.

15. Scott, W.B., "Experience From Desert Storm May Speed Missile Upgrades", Aviation Week & Space Technology, pp. 52-53, 22 July 1991.
16. Bruel & Kjaer (B&K), Instructions and Applications - Exciter Body Type 4801.
17. B&K, Instruction Manual - Vibration Exciter System V.
18. B&K, Instructions and Applications - General Purpose Head Type 4812.
19. B&K, Instruction Manual - Power Amplifier Type 2707.
20. PCB Piezoelectronics, Inc., Transducer Instrumentation Operating Instructions - FLEXCEL Accelerometer 336A Series.
21. PCB Piezoelectronics, Inc., Electronic Instrumentation Operating Instructions - Dual Mode Charge Amplifier Model 464A.
22. PCB Piezoelectronics, Inc., Transducer Instrumentation Operating Instructions - Impedance Head Model 288A11.

## BIBLIOGRAPHY

1. American Society of Mechanical Engineers (ASME), *Vibration Testing - Instrumentation and Data Analysis*, United Engineering Center, 1975.
2. Cook, R.D., Malkus, D.S., and Plesha, M.E., *Concepts and Applications of Finite Element Analysis*, 3<sup>rd</sup> ed., John Wiley & Sons, Inc., 1989.
3. Elishakoff, I. and Lyon, R.H., *Random Vibration - Status and Recent Developments*, Elsevier Science Publishers, 1986.
4. Harris, C.M., and Crede, C.E., *Shock and Vibration Handbook*, McGraw-Hill Book Co., 1976.
5. Naval Air Systems Command Technical Manual NAVAIR01-AIM9-2, *Sidewinder Guided Missile AIM-9H/L/M Organizational and Intermediate Maintenance With Illustrated Parts Breakdown*.
6. Naval Air Systems Command Technical Manual NAVAIR01-75PA-75, *Airborne Weapons/Stores Loading Manual, Navy Model P-3 Aircraft*, 15 April 1989.
7. Thomson, W.T., *Vibration Theory and Applications*, Prentice-Hall, Inc., 1965.
8. Tse, F.S., Morse, I.E., and Hinkle, R.T., *Mechanical Vibrations - Theory and Applications*, Allyn and Bacon, Inc., 1978.

# INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center 2  
Cameron Station  
Alexandria, VA 22304-6145
2. Library, Code 52 2  
Naval Postgraduate School  
Monterey, CA 93943-5002
3. Commander (AIR-5114B) 1  
Attn: CDR Boyington  
Naval Air Systems Command  
Washington, D.C. 23061-0001
4. Department Chairman 2  
Department of Aeronautics and Astronautics  
Naval Postgraduate School  
Monterey, CA 93943-5000
5. Commander, Naval Air Test Center 1  
Attn: Mr. Tim Twigg  
FWATD, Code FW52B  
NAS Patuxent River, MD 20670
6. Commander, Patrol Squadron 47 1  
Attn: LCDR R. Bobbitt  
FPO San Francisco, CA 96601-5920
7. Mr. Michael Shutty 3  
Naval Air Test Center  
FWATD, Code FW522MS  
NAS Patuxent River, MD 20670
8. Professor Edward Wu 3  
Department of Aeronautics and Astronautics  
Naval Postgraduate School  
Monterey, CA 93943-5000